Micro Synthetic Jets as Effective Actuator

Mehti Koklu
Old Dominion University

Follow this and additional works at: https://digitalcommons.odu.edu/mae_etds

Part of the Aerospace Engineering Commons, and the Mechanical Engineering Commons

Recommended Citation
Koklu, Mehti. "Micro Synthetic Jets as Effective Actuator" (2007). Doctor of Philosophy (PhD), Dissertation, Mechanical & Aerospace Engineering, Old Dominion University, DOI: 10.25777/8ye4-fr45 https://digitalcommons.odu.edu/mae_etds/70

This Dissertation is brought to you for free and open access by the Mechanical & Aerospace Engineering at ODU Digital Commons. It has been accepted for inclusion in Mechanical & Aerospace Engineering Theses & Dissertations by an authorized administrator of ODU Digital Commons. For more information, please contact digitalcommons@odu.edu.
ABSTRACT

MICRO SYNTHETIC JETS AS EFFECTIVE ACTUATORS

Mehti Koklu
Old Dominion University, 2007
Director: Dr. Oktay Baysal

Synthetic jets have previously been studied as actuators for external macroflow control and recently been proposed for internal microflow applications. Despite the wide variety of the potential applications of synthetic jet actuators, the majority of the studies have been done at macro scales. Furthermore, there has not been any design methodology that addresses the effectiveness of the micro synthetic jet actuators. Bearing these needs in mind, a micro synthetic jet configuration is considered in a microscale environment where $Kn$ number is less than 0.1 and more than 0.001. Flowfields are simulated by solving the compressible Navier-Stokes equations. The wall boundary conditions have been modified to accommodate the slip velocity and the temperature jump conditions encountered for this specific range of the Knudsen numbers. The membrane motion is modeled in a realistic manner as a moving boundary in order to accurately compute the flow inside the actuator cavity.

Due to lack of experimental studies on micro synthetic jets, validation of the current method is accomplished in two steps. In the first step, capabilities of the methodology are tested successfully by computing flowfields inside a microchannel, microfilter, and micro backward facing step. In the second step, a realistic modeling of a synthetic jet in macro flow conditions is validated with experimental results.
As the main contribution of this study, a detailed parametric study is presented that covers a large design space of synthetic jet actuation and design variables. In this study, both the synthetic jets in quiescent environment and in cross flow conditions are considered. The design variables for the parametric study are the membrane oscillation frequency, the membrane oscillation amplitude, the orifice width, the orifice height, the cavity height, and the cavity width. Studying the characteristic length allows an understanding of a synthetic jet for different Knudsen and Reynolds numbers. The momentum flux, jet velocity, vortex formation and shedding, the area and the circulation of the vortex, are the metrics considered to determine the effectiveness of a synthetic jet.

The final phase of the present study is on developing and demonstrating a design optimization methodology. This is accomplished in two steps. First, each design variable is considered one at a time as and other design variables are kept constant. This approach yields an effective actuator when considering the possibility of the limits on any design variable to be constant. As compared to the baseline case, the optimization studies yield 2%, 15%, 15%, 200% increase in actuation efficiency when the single-variable is the orifice width, the orifice height, the cavity height, or the frequency, respectively. Then a multi variable optimization is performed to obtain the synthetic jet configuration that yields the best efficiency. This study includes shape optimization using shape parameters and Bezier polynomials. As compared to the baseline case, the shape optimization using shape parameters results in a 170% increase in the actuation efficiency while the shape optimization with Bezier polynomials results in more than 10 times increase.
# TABLE OF CONTENTS

## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table of Contents</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF TABLES</td>
<td>viii</td>
</tr>
</tbody>
</table>

## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Table of Contents</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF FIGURES</td>
<td>x</td>
</tr>
</tbody>
</table>

## NOMENCLATURE

<table>
<thead>
<tr>
<th>Table of Contents</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>NOMENCLATURE</td>
<td>xvii</td>
</tr>
</tbody>
</table>

## Section

1. **INTRODUCTION** ........................................................................................................1
   1.1. Motivation........................................................................................................1
   1.2. Flow Control.....................................................................................................4
   1.3. Micro Electro Mechanical Systems (MEMS)...................................................6
   1.4. Optimization.....................................................................................................10
   1.5. Shape Optimization.........................................................................................13
   1.6. Synthetic Jet Actuators................................................................................14
   1.7. Literature Review
       1.7.1. Characterization..................................................................................16
       1.7.2. Modeling...............................................................................................22
       1.7.3. Applications.........................................................................................24
       1.7.4. Synthetic Jet Optimization.................................................................29
   1.8. Outline............................................................................................................32
   1.9. Objectives.......................................................................................................33
   1.10. Contribution of the Research.........................................................................33

2. **MATHEMATICAL MODEL** ....................................................................................35
   2.1. Flow Modeling and Analysis...........................................................................35
       2.1.1. Governing Equations.............................................................................35
       2.1.2. Time Advancement...............................................................................36
       2.1.3. Spatial Discretization..........................................................................36
   2.2. Initial and Boundary Conditions....................................................................37
   2.3. Preconditioning..............................................................................................41
   2.4. Physics of Microflows....................................................................................43
       2.4.1. Flow Regimes.........................................................................................46
       2.4.2. Flow Modeling.......................................................................................48
       2.4.3. Governing Equations and Boundary Conditions.....................................54
   2.5. Optimization Methodology..............................................................................57
3. COMPUTATIONAL TOOLS ................................................................. 65
   3.1. Geometry and Meshing ................................................................. 65
   3.2. Flow Solver ............................................................................. 67
   3.3. Optimization Code ................................................................. 69
   3.4. Computational Resources ..................................................... 71

4. VALIDATION STUDIES ............................................................... 72
   4.1. Validation of Microflow Modeling ........................................... 72
       4.1.1. Straight Microchannel .................................................... 72
       4.1.2. Micro Filter .................................................................. 76
       4.1.3. Micro Backward Facing Step ......................................... 79
   4.2. Validation of Synthetic Jet Modeling ....................................... 81
       4.2.1. Synthetic Jet in a Quiescent Environment ....................... 81
       4.2.2. Synthetic Jet in Cross Flow ........................................... 86

5. SYNTHETIC JET IN A QUISCENT ENVIRONMENT ..................... 94
   5.1. Characteristic Length Scale .................................................... 105
   5.2. Characteristic Length Scale at Constant $k_r$ ......................... 113
   5.3. Amplitude ........................................................................... 116
   5.4. Frequency ........................................................................... 119
   5.5. Orifice Width ....................................................................... 123
   5.6. Orifice Height ..................................................................... 127
   5.7. Cavity Height ...................................................................... 131
   5.8. Cavity Width ...................................................................... 134
   5.9. Summary ............................................................................ 137

6. SYNTHETIC JET IN A CROSS FLOW .......................................... 139
   6.1. Characteristic Length Scale .................................................... 147
   6.2. Characteristic Length Scale at Constant $k_r$ ......................... 152
   6.3. Amplitude ........................................................................... 154
   6.4. Frequency ........................................................................... 158
   6.5. Orifice Width ....................................................................... 164
   6.6. Orifice Height ..................................................................... 169
   6.7. Cavity Height ...................................................................... 173
   6.8. Cavity Width ...................................................................... 178
   6.9. Incoming Velocity of Cross Flow ........................................... 182
   6.10. Summary ............................................................................ 187
# List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 2.1 Flow regimes</td>
<td>46</td>
</tr>
<tr>
<td>Table 5.1 Baseline case for quiescent case</td>
<td>99</td>
</tr>
<tr>
<td>Table 5.2 Tracking of vortex properties for quiescent environment</td>
<td>104</td>
</tr>
<tr>
<td>Table 5.3 Vortex properties table for ( I_c ) study</td>
<td>111</td>
</tr>
<tr>
<td>Table 5.4 Vortex properties table for ( k_r ) study</td>
<td>115</td>
</tr>
<tr>
<td>Table 5.5 Vortex properties table for amplitude study</td>
<td>117</td>
</tr>
<tr>
<td>Table 5.6 Vortex properties table for frequency study</td>
<td>121</td>
</tr>
<tr>
<td>Table 5.7 Vortex properties table for orifice width study</td>
<td>125</td>
</tr>
<tr>
<td>Table 5.8 Vortex properties table for orifice height study</td>
<td>128</td>
</tr>
<tr>
<td>Table 5.9 Vortex properties table for cavity height study</td>
<td>132</td>
</tr>
<tr>
<td>Table 5.10 Vortex properties table for cavity width study</td>
<td>135</td>
</tr>
<tr>
<td>Table 6.1 Baseline case for cross flow condition</td>
<td>141</td>
</tr>
<tr>
<td>Table 6.2 Tracking of vortex properties of one vortex</td>
<td>143</td>
</tr>
<tr>
<td>Table 6.3 Vortex properties table for ( I_c ) study</td>
<td>149</td>
</tr>
<tr>
<td>Table 6.4 Vortex properties table for ( k_r ) study</td>
<td>153</td>
</tr>
<tr>
<td>Table 6.5 Vortex properties table for amplitude study</td>
<td>156</td>
</tr>
<tr>
<td>Table 6.6 Vortex properties table of first vortex for frequency study</td>
<td>161</td>
</tr>
<tr>
<td>Table 6.7 Vortex properties table of second vortex for frequency study</td>
<td>161</td>
</tr>
<tr>
<td>Table 6.8 Vortex properties table for orifice width study</td>
<td>166</td>
</tr>
<tr>
<td>Table 6.9 Vortex properties table of first vortex for orifice height study</td>
<td>171</td>
</tr>
<tr>
<td>Table</td>
<td>Page</td>
</tr>
<tr>
<td>--------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Table 6.10 Vortex properties table of second vortex for orifice height study</td>
<td>171</td>
</tr>
<tr>
<td>Table 6.11 Vortex properties table of first vortex for cavity height study</td>
<td>175</td>
</tr>
<tr>
<td>Table 6.12 Vortex properties table of second vortex for cavity height study</td>
<td>175</td>
</tr>
<tr>
<td>Table 6.13 Vortex properties table of first vortex for cavity width study</td>
<td>180</td>
</tr>
<tr>
<td>Table 6.14 Vortex properties table of second vortex for cavity width study</td>
<td>180</td>
</tr>
<tr>
<td>Table 6.15 Vortex properties table for incoming flow study</td>
<td>185</td>
</tr>
<tr>
<td>Table 7.1 Case table for orifice width optimization</td>
<td>191</td>
</tr>
<tr>
<td>Table 7.2 Case table for cavity height optimization</td>
<td>194</td>
</tr>
<tr>
<td>Table 7.3 Case table for orifice height optimization</td>
<td>197</td>
</tr>
<tr>
<td>Table 7.4 Case table for frequency optimization</td>
<td>208</td>
</tr>
<tr>
<td>Table 7.5 Case table for shape optimization with shape parameters</td>
<td>214</td>
</tr>
<tr>
<td>Table 7.6 Side constraints for shape optimization</td>
<td>221</td>
</tr>
<tr>
<td>Table 7.7 Case table for shape optimization with Bezier curves</td>
<td>221</td>
</tr>
<tr>
<td>Table 7.8 Comparison of obtained objective function values</td>
<td>226</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1 Principle of a synthetic jet</td>
<td>15</td>
</tr>
<tr>
<td>Figure 2.1 Effect of preconditioning on the convergence [97]</td>
<td>42</td>
</tr>
<tr>
<td>Figure 2.2 Flow modeling classifications</td>
<td>49</td>
</tr>
<tr>
<td>Figure 2.3 Optimization process flowchart</td>
<td>61</td>
</tr>
<tr>
<td>Figure 2.4 Definition of active-violated constraints</td>
<td>62</td>
</tr>
<tr>
<td>Figure 4.1 Computational grid for a straight microchannel</td>
<td>73</td>
</tr>
<tr>
<td>Figure 4.2 The centerline pressure distribution through the microchannel</td>
<td>74</td>
</tr>
<tr>
<td>Figure 4.3 Deviation of centerline pressure from the analytical solution</td>
<td>74</td>
</tr>
<tr>
<td>Figure 4.4 Variation of slip velocity along the microchannel wall</td>
<td>75</td>
</tr>
<tr>
<td>Figure 4.5 Characteristic dimensions of the micro filter</td>
<td>76</td>
</tr>
<tr>
<td>Figure 4.6 Micro filter and its computational domain</td>
<td>77</td>
</tr>
<tr>
<td>Figure 4.7 Normalized centerline velocity variations</td>
<td>78</td>
</tr>
<tr>
<td>Figure 4.8 Normalized centerline temperature variations</td>
<td>78</td>
</tr>
<tr>
<td>Figure 4.9 Computational grid for backward facing step</td>
<td>79</td>
</tr>
<tr>
<td>Figure 4.10 Comparison of centerline pressure variation</td>
<td>80</td>
</tr>
<tr>
<td>Figure 4.11 Comparison of centerline velocity variation</td>
<td>81</td>
</tr>
<tr>
<td>Figure 4.12 Physical domain of synthetic jet actuator [41]</td>
<td>82</td>
</tr>
<tr>
<td>Figure 4.13 Computational grid for synthetic jet actuator</td>
<td>83</td>
</tr>
<tr>
<td>Figure 4.14 Displacement of membrane in one cycle</td>
<td>84</td>
</tr>
<tr>
<td>Figure 4.15 Comparison to the experimental results for quiescent case</td>
<td>86</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>4.16</td>
<td>Perspective of hump model</td>
</tr>
<tr>
<td>4.17</td>
<td>Computational grid of the hump model</td>
</tr>
<tr>
<td>4.18</td>
<td>Comparison of two turbulence models with experiment</td>
</tr>
<tr>
<td>4.19</td>
<td>Effect of preconditioning</td>
</tr>
<tr>
<td>4.20</td>
<td>Comparison to experimental results for no flow control case</td>
</tr>
<tr>
<td>4.21</td>
<td>Comparison of results with experimental data for flow control case</td>
</tr>
<tr>
<td>4.22</td>
<td>Results of Viken, S. from NASA LARC</td>
</tr>
<tr>
<td>4.23</td>
<td>Results of Duraisamy et al. from University of Maryland</td>
</tr>
<tr>
<td>4.24</td>
<td>Results of Krishnan et al. from University of Arizona</td>
</tr>
<tr>
<td>5.1</td>
<td>Schematics of synthetic jet operation</td>
</tr>
<tr>
<td>5.2</td>
<td>Synthetic jet membrane and its maximum deflection</td>
</tr>
<tr>
<td>5.3</td>
<td>Limit cycle for mass flow rates</td>
</tr>
<tr>
<td>5.4</td>
<td>Numerical grid for the synthetic jet configuration</td>
</tr>
<tr>
<td>5.5</td>
<td>Vorticity contours at four stages of one oscillation cycle</td>
</tr>
<tr>
<td>5.6</td>
<td>Tracking of a vortex in a quiescent environment</td>
</tr>
<tr>
<td>5.7</td>
<td>Velocity profiles at four stages of one oscillation cycle</td>
</tr>
<tr>
<td>5.8</td>
<td>Vorticity contours at four stages of one oscillation cycle for lc=500 μm</td>
</tr>
<tr>
<td>5.9</td>
<td>Variation of the vortex area with respect to time</td>
</tr>
<tr>
<td>5.10</td>
<td>Variation of circulation with time</td>
</tr>
<tr>
<td>5.11</td>
<td>Variation of unit circulation with time</td>
</tr>
<tr>
<td>5.12</td>
<td>Variation of maximum vorticity with time</td>
</tr>
<tr>
<td>5.13</td>
<td>Distance traveled by the vortex</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>Figure 5.14 Plot of vorticity contours for three different characteristic lengths</td>
<td>111</td>
</tr>
<tr>
<td>Figure 5.15 Effect of characteristic length on orifice jet velocity profile</td>
<td>112</td>
</tr>
<tr>
<td>Figure 5.16 Variation of momentum flux with respect to characteristic length</td>
<td>113</td>
</tr>
<tr>
<td>Figure 5.17 Plot of vorticity contours at different characteristic lengths</td>
<td>115</td>
</tr>
<tr>
<td>Figure 5.18 Variation of momentum flux at constant $k_r$</td>
<td>116</td>
</tr>
<tr>
<td>Figure 5.19 Plot of vorticity contours for different amplitudes</td>
<td>117</td>
</tr>
<tr>
<td>Figure 5.20 Effect of amplitude on velocity profiles at the orifice exit</td>
<td>118</td>
</tr>
<tr>
<td>Figure 5.21 Variation of momentum flux with respect to membrane amplitude</td>
<td>119</td>
</tr>
<tr>
<td>Figure 5.22 Plot of vorticity contours for different frequencies</td>
<td>120</td>
</tr>
<tr>
<td>Figure 5.23 Effect of frequency on velocity profiles at the orifice exit</td>
<td>122</td>
</tr>
<tr>
<td>Figure 5.24 Variation of momentum flux with respect to frequency</td>
<td>123</td>
</tr>
<tr>
<td>Figure 5.25 Plot of vorticity contours for different orifice widths</td>
<td>124</td>
</tr>
<tr>
<td>Figure 5.26 Effect of orifice width on velocity profiles at the orifice exit</td>
<td>126</td>
</tr>
<tr>
<td>Figure 5.27 Variation of momentum flux with respect to orifice width</td>
<td>127</td>
</tr>
<tr>
<td>Figure 5.28 Plot of vorticity contours for different orifice heights</td>
<td>128</td>
</tr>
<tr>
<td>Figure 5.29 Effect of orifice height on velocity profiles at the orifice exit</td>
<td>130</td>
</tr>
<tr>
<td>Figure 5.30 Variation of momentum flux with respect to orifice height</td>
<td>131</td>
</tr>
<tr>
<td>Figure 5.31 Plot of vorticity contours for different cavity heights</td>
<td>132</td>
</tr>
<tr>
<td>Figure 5.32 Effect of cavity height on velocity profiles at the orifice exit</td>
<td>133</td>
</tr>
<tr>
<td>Figure 5.33 Variation of momentum flux with respect to cavity height</td>
<td>133</td>
</tr>
<tr>
<td>Figure 5.34 Plot of vorticity contours for different cavity width</td>
<td>135</td>
</tr>
<tr>
<td>Figure 5.35 Effect of cavity width on velocity profiles at the orifice exit</td>
<td>136</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>5.36</td>
<td>Variation of momentum flux with respect to cavity width</td>
</tr>
<tr>
<td>5.37</td>
<td>Summary of the quiescent case results</td>
</tr>
<tr>
<td>6.1</td>
<td>Computational grid for cross flow case</td>
</tr>
<tr>
<td>6.2</td>
<td>Vorticity contours at four stages of membrane oscillation cycle</td>
</tr>
<tr>
<td>6.3</td>
<td>Tracking of single vortex in cross flow</td>
</tr>
<tr>
<td>6.4</td>
<td>Velocity profiles at four stages of membrane oscillation cycle</td>
</tr>
<tr>
<td>6.5</td>
<td>Plot of vorticity contours for different characteristic lengths</td>
</tr>
<tr>
<td>6.6</td>
<td>Effect of characteristic length on velocity profiles at the orifice exit</td>
</tr>
<tr>
<td>6.7</td>
<td>Detail look up to orifice at first stage</td>
</tr>
<tr>
<td>6.8</td>
<td>Variation of momentum flux with respect to characteristic length</td>
</tr>
<tr>
<td>6.9</td>
<td>Plot of vorticity contours for different characteristic lengths</td>
</tr>
<tr>
<td>6.10</td>
<td>Variation of momentum flux with respect to characteristic length</td>
</tr>
<tr>
<td>6.11</td>
<td>Plot of vorticity contours for different amplitudes</td>
</tr>
<tr>
<td>6.12</td>
<td>Effect of amplitude on velocity profiles at the orifice exit</td>
</tr>
<tr>
<td>6.13</td>
<td>Variation of momentum flux with respect to amplitude</td>
</tr>
<tr>
<td>6.14</td>
<td>Plot of vorticity contours for different frequencies</td>
</tr>
<tr>
<td>6.15</td>
<td>Plot of vorticity contours at four stages for, f=30, 50, 70 kHz</td>
</tr>
<tr>
<td>6.16</td>
<td>Effect of frequency on velocity profiles at the orifice exit</td>
</tr>
<tr>
<td>6.17</td>
<td>Variation of momentum flux with respect to frequency</td>
</tr>
<tr>
<td>6.18</td>
<td>Plot of vorticity contours for different orifice widths</td>
</tr>
<tr>
<td>6.19</td>
<td>Effect of orifice width on velocity profiles at the orifice exit</td>
</tr>
<tr>
<td>6.20</td>
<td>Variation of momentum flux with respect to orifice width</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>Figure 6.21 Plot of vorticity contours for different orifice heights</td>
<td>170</td>
</tr>
<tr>
<td>Figure 6.22 Effect of orifice height on velocity profiles at the orifice exit</td>
<td>172</td>
</tr>
<tr>
<td>Figure 6.23 Variation of momentum flux with respect to orifice height</td>
<td>173</td>
</tr>
<tr>
<td>Figure 6.24 Plot of vorticity contours for different cavity heights</td>
<td>174</td>
</tr>
<tr>
<td>Figure 6.25 Effect of cavity height on velocity profiles at the orifice exit</td>
<td>177</td>
</tr>
<tr>
<td>Figure 6.26 Variation of momentum flux with respect to cavity height</td>
<td>178</td>
</tr>
<tr>
<td>Figure 6.27 Plot of vorticity contours for different cavity widths</td>
<td>179</td>
</tr>
<tr>
<td>Figure 6.28 Effect of cavity width on velocity profiles at the orifice exit</td>
<td>181</td>
</tr>
<tr>
<td>Figure 6.29 Variation of momentum flux with respect to cavity width</td>
<td>182</td>
</tr>
<tr>
<td>Figure 6.30 Plot of vorticity contours for different incoming flow velocities</td>
<td>183</td>
</tr>
<tr>
<td>Figure 6.31 Effect of incoming velocity on velocity profiles at the orifice exit</td>
<td>186</td>
</tr>
<tr>
<td>Figure 6.32 Variation of momentum flux with respect to incoming flow velocity</td>
<td>187</td>
</tr>
<tr>
<td>Figure 6.33 Summary of the cross flow case</td>
<td>188</td>
</tr>
<tr>
<td>Figure 7.1 Optimization history of orifice width optimization for case 1</td>
<td>191</td>
</tr>
<tr>
<td>Figure 7.2 Optimization history of orifice width optimization for case 2</td>
<td>192</td>
</tr>
<tr>
<td>Figure 7.3 Design variable history of orifice width optimization for both cases</td>
<td>192</td>
</tr>
<tr>
<td>Figure 7.4 Objective function history of orifice width optimization for both cases</td>
<td>193</td>
</tr>
<tr>
<td>Figure 7.5 Optimization history of cavity height optimization for case 1</td>
<td>195</td>
</tr>
<tr>
<td>Figure 7.6 Optimization history of cavity height optimization for case 2</td>
<td>195</td>
</tr>
<tr>
<td>Figure 7.7 Design variable history of cavity height optimization for both cases</td>
<td>196</td>
</tr>
<tr>
<td>Figure 7.8 Objective function history of cavity height optimization for both cases</td>
<td>196</td>
</tr>
<tr>
<td>Figure 7.9 Design variable history of orifice height optimization</td>
<td>198</td>
</tr>
</tbody>
</table>
Figure 7.10 Objective function history of orifice height optimization ........................................ 198
Figure 7.11 Variation of momentum flux with respect to frequency ........................................ 201
Figure 7.12 Momentum variation at different characteristic lengths ........................................ 201
Figure 7.13 Momentum Variations for 25 and 50 μm ............................................................. 202
Figure 7.14 Original points and related data fitted curve ....................................................... 205
Figure 7.15 Original and the improved data points and their fitted curves ............................. 207
Figure 7.16 Optimization history of frequency optimization for Case 1 .............................. 208
Figure 7.17 Optimization history of frequency optimization for Case 2 .............................. 209
Figure 7.18 Design variable history for frequency optimization for both cases .................. 209
Figure 7.19 Objective function history for frequency optimization for both cases .......... 210
Figure 7.20 Sample Bezier curves ........................................................................................ 213
Figure 7.21 Optimization history for St .................................................................................... 215
Figure 7.22 Optimization history for design variable (d₀) ..................................................... 216
Figure 7.23 Optimization history for design variable (h₀) ..................................................... 216
Figure 7.24 Optimization history for design variable (H) .................................................... 217
Figure 7.25 Optimization history of the objective function ................................................... 218
Figure 7.26 Cavity shape with Bezier curves ........................................................................... 219
Figure 7.27 Optimization history of the nondimensional St constraint ................................ 222
Figure 7.28 Optimization histories of design variables ......................................................... 223
Figure 7.29 Optimization history of design variable (x₃) ....................................................... 224
Figure 7.30 Optimization history of the objective function .................................................. 224
Figure 7.31 Baseline, initial and optimum synthetic jet shape ............................................... 225
Figure B.1 Initial grid created by a simple FORTRAN code ........................................... 262
Figure B.2 First obtained grid after the optimization run ............................................ 264
Figure B.3 Final grid zoomed inside the left corner ...................................................... 265
## NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Membrane oscillation amplitude</td>
</tr>
<tr>
<td>$a$</td>
<td>Speed of sound</td>
</tr>
<tr>
<td>$c$</td>
<td>Sutherland’s constant</td>
</tr>
<tr>
<td>$C_q$</td>
<td>Mass flow</td>
</tr>
<tr>
<td>$C_p$, $C_v$</td>
<td>Specific heats</td>
</tr>
<tr>
<td>$\bar{C}$</td>
<td>Vorticity cut off value</td>
</tr>
<tr>
<td>$d$</td>
<td>Molecular diameter</td>
</tr>
<tr>
<td>$d_o$</td>
<td>Orifice width</td>
</tr>
<tr>
<td>$d^k$</td>
<td>Search direction</td>
</tr>
<tr>
<td>$d_{yx}$</td>
<td>Sharpness factor of the left edge</td>
</tr>
<tr>
<td>$d_{min}$</td>
<td>Minimum gap at the orifice</td>
</tr>
<tr>
<td>$dE_i$, $dE_r$</td>
<td>Incoming and reflected energy fluxes of molecules per unit time</td>
</tr>
<tr>
<td>$e$, $E$</td>
<td>Internal, total specific energy</td>
</tr>
<tr>
<td>$Ec$</td>
<td>Eckert number</td>
</tr>
<tr>
<td>$f$</td>
<td>Membrane oscillation frequency</td>
</tr>
<tr>
<td>$f_o$</td>
<td>Maxwellian distribution function</td>
</tr>
<tr>
<td>$\bar{f}$</td>
<td>External forces acting on the control volume</td>
</tr>
<tr>
<td>$\bar{f}$</td>
<td>Velocity distribution function</td>
</tr>
<tr>
<td>FD</td>
<td>Nondimensional frequency</td>
</tr>
</tbody>
</table>
\( \hat{F}_v, \hat{G}_v, \hat{H}_v \) Viscous flux terms

\( \hat{F}, \hat{G}, \hat{H} \) Inviscid flux terms

\( g_i \) Inequality constraints

\( H \) Cavity height

\( \bar{H} \) Approximated Hessian of the Lagrangian function

\( h_c \) Channel height

\( h_o \) Orifice height

\( h_j \) Equality constraints

\( \bar{I} \) Unit tensor

\( J \) Jacobian of coordinate transformation

\( \bar{J} \) Objective function

\( k \) Thermal conductivity

\( Kn \) Knudsen number

\( k_r \) Reduced frequency

\( l_c \) Characteristic length

\( m \) Control points

\( \dot{m} \) Mass flow rate

\( M \) Mach number

\( \dot{M} \) Total mass flow rate

\( n \) Number density of the molecules

\( \bar{n} \) Outward normal unit vector

\( N_{MX}^L \) Number of local maximums
\( p \) Pressure

\( P \) Momentum flux at the orifice exit \((kg.m/s^2)\)

\( \bar{\rho} \) Local preconditioning matrix

Pr Prandtl number

\( \bar{q} \) Heat flux vector

\( \dot{Q} \) Vector of variables, (density, momentum and total energy)

\( q_i(x) \) Quality metrics

\( q_{in} \) Inlet dynamic head

\( q_n, q_s \) Normal and tangential heat-flux components

\( R \) Universal gas constant

Re Reynolds number

\( \text{Re}_c \) Reynolds number based on chord length

\( \bar{R} \) Residual of a function

\( R^- , R^+ \) Riemann invariants

\( r_{xy} \) Sharpness factor of orifice lip

\( S \) Dimensionless vortex area

\( s \) Step height

\( \tilde{s} \) Tangential unit vector

\( T \) Temperature

\( \tilde{T}_w \) Wall temperature

\( U \) Velocity

\( U_\infty \) Free stream velocity
\( \overline{U} \) Normalized contravariant velocity
\( \overline{u} \) Instantaneous velocity at the orifice exit
\( V \) Spatially-averaged total fluid velocity during expulsion stage
\( V_{\text{jet}} \) Fluctuating jet velocity
\( V_{\text{max}}, V_{\text{min}} \) Maximum and minimum velocities
\( W \) Cavity width
\( x, y \) Cartesian coordinate system
\( x^L, x^U \) Side constraints for design variables
\( x_L, x_R \) New design points for frequency variation

**Greek Symbols**
\( \alpha \) Step size
\( \beta \) Opening factor
\( \gamma \) Ideal gas constant
\( \Gamma \) Dimensionless circulation
\( \Gamma/S \) Dimensionless unit circulation
\( \theta_j \) Push-off factor
\( \bar{\theta} \) Angle constraint
\( \lambda \) Mean free path
\( \mu_i, \lambda_j \) Lagrange multipliers
\( \nu, \mu \) Kinematic and dynamic viscosities
\( \xi \) Vorticity
\( \xi_{\text{max}} \) Maximum vorticity
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\xi_{\text{loc}}$</td>
<td>Maximum vorticity location</td>
</tr>
<tr>
<td>$(\xi, \eta, \zeta)$</td>
<td>Generalized coordinates</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Normal stress tensor</td>
</tr>
<tr>
<td>$\sigma_v$</td>
<td>Momentum accommodation coefficients</td>
</tr>
<tr>
<td>$\sigma_T$</td>
<td>Thermal accommodation coefficients</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Viscous stress tensor</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>Control volume, area of computational domain</td>
</tr>
<tr>
<td>$\partial \Omega$</td>
<td>Control surface</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Stage</td>
</tr>
<tr>
<td>$\nabla^2$</td>
<td>Laplacian</td>
</tr>
<tr>
<td>$\nabla$</td>
<td>Divergence</td>
</tr>
<tr>
<td>$^0$</td>
<td>Initial and optimal values</td>
</tr>
</tbody>
</table>

**Abbreviations**

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANOVA</td>
<td>Analysis Of Variance</td>
</tr>
<tr>
<td>BFGS</td>
<td>Broydon-Fletcher-Goldfarb-Shanno</td>
</tr>
<tr>
<td>CFL3D</td>
<td>Computational Fluids Laboratory 3-Dimensional</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulations</td>
</tr>
<tr>
<td>DSMC</td>
<td>Direct Simulation Of Monte Carlo</td>
</tr>
<tr>
<td>KKT</td>
<td>Karush Kuhn Tucker</td>
</tr>
<tr>
<td>MEMS</td>
<td>Micro Elektro Mechanical Systems</td>
</tr>
<tr>
<td>MESQUITE</td>
<td>Mesh Quality Improvement Toolkit</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>-----------------------------------------------</td>
</tr>
<tr>
<td>MD</td>
<td>Molecular Dynamics</td>
</tr>
<tr>
<td>LDV</td>
<td>Laser Doppler Velocimetry</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle Image Velocimetry</td>
</tr>
<tr>
<td>SA</td>
<td>Splarat Almaras turbulence model</td>
</tr>
<tr>
<td>SQP</td>
<td>Sequential Quadratic Programming</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport turbulence model</td>
</tr>
<tr>
<td>TFI</td>
<td>Trans-Finite Interpolation</td>
</tr>
</tbody>
</table>
1. INTRODUCTION

1.1. Motivation

Manipulating a flow field to affect a desired change is of enormous technological importance, and this unquestionably explains the fact that the subject is more intensely followed by scientists and engineers than any other topic in fluid mechanics. To reach this aim, numerous methods, ideas, and devices have been proposed and applied in this challenging area. As the progress in the MEMS technology and its micromachining techniques continues, the increased performance and reliability of MEMS devices fascinated the flow control researchers. Furthermore, the obvious advantages of the reduced physical size, volume, weight and cost attracts many flow control researchers to the MEMS field. Likewise, in many research areas, the MEMS devices have spread over the flow control applications. One such device, namely, the synthetic jet actuators are very popular for external flow applications in macroscale and have recently been proposed as versatile flow control devices for micro-electro-mechanical systems (MEMS) applications (Aslan et al. [1]).

The potential applications of the synthetic jet actuators are diverse and technologically significant. They include boundary layer and separation control, mixing enhancement and microchip cooling. In addition, the synthetic jet actuators offer excellent spatial resolution and high operational frequency with low power consumption. Despite the wide variety of potential applications of the synthetic jet actuators, there has not been any design methodology, which addresses the effectiveness of the actuators.
More generally, there has not been an answer to the question how do we design the synthetic jet actuators in order to get the best performance? Or how do we know that we are on the bad side of the design space?

Despite the wide variety of the potential applications of the synthetic jet actuators, the majority of the recent studies about the synthetic jets are at macro scales. As the MEMS technology makes significant progress, these previous macroscale works have to be carried out into this new fascinating world. First of all, a detailed parametric study that spans the whole applicable range of the synthetic jet is a must for an extensive effectiveness study. Even though it is possible to enumerate some previous parametric studies, most of them have suffered from insufficient design points.

After a detailed parametric study, the next step should be a design optimization study. The design optimization study is needed because it reduces cost and turnaround time in developing microfluidic systems. In addition, when using the design optimization the product quality and performance increase. However, MEMS design process is usually performed in a trial-and-error fashion. This is a non-ideal, non-scientific design methodology because it involves a long time due to the prohibitive number of iteration requirements. This time consuming and non-scientific methodology reflects very high cost for commercial product development. On the contrary, there have been only a few in depth investigations of optimization of the synthetic jets in macro scale yet the optimization of a micro synthetic jet in microflows has never been touched. An optimally designed and actuated synthetic jet should lead to a synthetic jet that can be designed effectively at a lower cost with fewer power requirements as well as higher performance.
In the literature, there have been numerous studies and a couple of patents on these types of actuators. However, this study has completely different objectives and methodology. The main difference is that we are studying micron-scale flows where in the previous studies micro refers to the fabrication technique not the device size of flow physics. In our study, the Knudsen number is less than 0.1 and more than 0.001. In this Knudsen number range, flow is in the slip flow regime. In case of slip flow, the mass flow, the shear stress, and the separated flow characteristics are different from no-slip flow counterparts. Therefore, device design and shapes need to consider relevant physics.

The other important difference is that the primary focus of our design is on the analysis of the design space determined by the geometric and flow-type design variables that identify the effectiveness of the micro synthetic jet actuator. The present results for jet discharging into quiescent environment and cross flow conditions reveal that these variables have determining effects on the synthetic jet effectiveness metrics, which are the jet exit velocity, the momentum flux, and the vortex properties.

The other motivating issue is the availability of the synthetic jets to technology transfer rather than being a theoretical concept. There are overestimated numbers of patent applications that are already applied to technology transfer (Glezer et al. [2], [3], [4], [5]). As the electronic products become more sophisticated and design margins tighten, micro synthetic jets are taking part in the innovative flow control strategy world. Cooling of micro electronic components is an especially hot topic nowadays. Some commercial companies have already been founded to enable new electronic design through sophisticated cooling technology using the synthetic jets [6].

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
1.2. Flow Control

In many fluid flows, numerous adverse flow conditions arise such as flow separation, vortex breakdown, shock waves, etc. These adverse effects lead to an increase in drag and in other losses and are needed to be sensed, manipulated, and controlled. The manipulation of a flow field to effect a desired change is of enormous technological importance, and this unquestionably explains the fact that the subject is more intensely followed by scientists and engineers than any other topic in fluid mechanics. Flow control is the effort to favorably modify the character or nature of a flow field. The potential benefits of flow control include enhanced performance and maneuverability, affordability, increased range and payload, and environmental compliance. More generally, the aim of flow control is to alleviate these adverse effects and increase the performance.

Flow control involves passive or active devices to effect a beneficial change in wall bounded or free-shear flows. The intent of flow control may be to delay/advance transition, to suppress/enhance turbulence, or to prevent/promote separation. The resulting benefits include drag reduction, lift enhancement, mixing augmentation, heat transfer enhancement, and flow-induced noise suppression.

It is widely accepted that Prandtl (1904) pioneered the modern use of flow control in his presentation. He introduced for the first time the boundary layer theory, explained the mechanics of steady separation, and described several experiments in which the boundary layer was controlled. In his study, Prandtl used suction to delay the boundary layer separation from the surface of the cylinder. From then on the military as well as industrial private corporations invested valuable resources searching for new methods to
conserve energy and hence reduce drag for air, sea, and land vehicles, for pipelines and for other industrial devices.

Categorization of flow control methods is based on energy expenditure [7]. Flow control involves passive or active devices that have a beneficial change on the flow field. A considerable amount of research has been performed using passive methods of flow control that modify a flow without external energy expenditure. Passive techniques include geometric shaping to manipulate the pressure gradient, the use of fixed mechanical vortex generators for separation control, and placement of longitudinal grooves or riblets on a surface to reduce drag. Recent reviews of passive flow control include Bushnell and Hefner [8] and Gad-el-Hak [9]. A detailed historical perspective on flow control is also provided by Gad-el-Hak [9]. During the last decade, emphasis has been on the development of active control methods in which energy or auxiliary power is introduced into the flow. Examples of active flow control include jet vectoring using piezoelectric actuators [10] and post-stall lift enhancement and drag reduction using oscillatory blowing (Seifert and Pack, [11]). In general, the categorization of the flow control can be tabulated as follows:
• Passive Control:
  • Riblets
  • Vortex generators
  • Passive cavities

• Active Control:
  • Deforming surfaces
  • Pulsed jets
  • Active suction
  • MEMS actuators:
    – The micro-flap
    – The cantilever cavity actuator
    – Synthetic jet actuator

Of all these active-passive control methods, MEMS actuators have the greatest potential.

1.3. Micro Electro Mechanical Systems (MEMS)

During the past decade, micromachining technology has become available to fabricate micron-sized mechanical parts. Micro electromechanical systems (MEMS) generally range in size from a micrometer (a millionth of a meter) to a millimeter (a thousandth of a meter). MEMS combine both electrical and mechanical components, and they are fabricated using integrated circuit batch-processing technologies. Micro devices may have the characteristics length smaller than the diameter of a human hair. At these size scales, the standard constructs of classical physics do not always hold true. This makes the micro machine field a new technology as well as a new scientific frontier. When dealing with flow configurations of microns or less, we have observed many unexpected phenomena. Due to MEMS large surface area to volume ratio, surface effects such as electrostatics and wetting dominate volume effects such as inertia or thermal mass. Even though the current manufacturing techniques for MEMS are out of the scope of this study, they include surface silicon micromachining; bulk silicon
micromachining; lithography, electro deposition and plastic molding (or, in its original German, Lithographie Galvanoformung Abformung, LIGA); and Electro Discharge Machining (EDM) ([9], [12], [13]).

The reason for the popularity of MEMS is due to its broad technology base building on expertise on silicon semiconductor manufacturing. In the first place, the interdisciplinary nature of MEMS technology and its micromachining techniques as well as its diversity of applications has resulted in an unprecedented range of devices and synergies across previously unrelated fields (for example, biology and microelectronics). Secondly, MEMS, with its batch fabrication techniques, enables components and devices to be manufactured with increased performance and reliability, combined with the obvious advantages of the reduced physical size, volume, weight, and cost. Thirdly, MEMS provides the basis for the manufacture of products that cannot be made by other methods. These factors make MEMS a potentially more pervasive technology than integrated circuit microchips.

Microfluidics, however, appeared as the name for the new research discipline of MEMS dealing with transport phenomena and fluid based devices at the microscopic length scale. Its process includes pumping, flow switching, mixing, incubating, sample dispensing, and molecule particle separating in microchannels. Recent microfluidics research has led to micro devices that can control macroscopic turbulence flows or flows in microchannels.

Their small size, fast response, low unit cost, minimal energy consumption, and an ability to combine mechanical and electronic components have presently made MEMS-based sensors and actuators the best candidate for active flow control.
The famed physicist Richard P. Feynman delivered two profound lectures on electromechanical miniaturization: “There’s Plenty of Room at the Bottom,” presented at the annual meeting of the American Physical Society, Pasadena, California, 29 December 1959, and “Infinitesimal Machinery,” presented at the Jet Propulsion Laboratory on 23 February 1983. Despite Feynman’s doubt regarding the usefulness of small machines, MEMS are finding increased applications in a variety of industrial and medical fields, with a potential worldwide market in the billions of dollars ($30 billion by 2004) [9]. The beginning of the micro machine field is signified by an electro-statically driven motor (Fan et al. [14]). A comb structure (Tang et al. [15]) derived from the micro motor concept eventually evolved into the airbag sensor, which reduces the damage caused by automobile collisions and is used now on almost all American-made cars.

From then on, numerous devices for actuating a variety of flow fields have been advanced during the last few years [12]. Wiltse and Glezer [16] originated an interesting path when they used piezoelectric-driven cantilevers as flow control actuators to beneficially affect free-shear layers. The following the idea of Wiltse and Glezer [16], Jacobson and Reynolds [17] designed an actuator that has the piezoelectric cantilever mounted flush with a boundary layer surface. The cantilever was oriented in the streamwise direction and fixed to the wall at its upstream end and could flex out into the flow and down into a cavity in the wall. The under cantilever cavity was filled with fluid that was connected to the flow through the gaps on the sides and at the tip of the cantilever. Hence, when the cantilever flexed into the cavity, it forced fluid out of the cavity into the external flow, and when the cantilever flexed out of the cavity, fluid was drawn into the cavity from the external flow. Neither Wiltse and Glezer [16] nor
Jacobson and Reynolds [17] used MEMS technology to fabricate their actuators, but the main conclusion drawn by Jacobson and Reynolds [17] was that future actuators must be MEMS-based in order to match the relevant scales of turbulent flows.

It is known that shear flows are sensitive to perturbations. If these perturbations are excited at the orifice of the shear layer and are within the instability band, the streamwise development of the flow can be significantly modified [18], [19]. This type of flow control scheme has been demonstrated to be very effective on free shear flow because it takes advantage of the flow instability that is a powerful amplifier. Amplification is especially important because these actuators cannot deliver high power or large forces.

For the turbulent wall bounded shear flow, there are different types of challenges and opportunities. The flow structures responsible for various drag increases in turbulent flow are very small in size typically several hundred microns in width and several mm in length. Their lifetime is short -in the millisecond range or less. Great difficulties arise from the fact that they are randomly distributed in time and space. Direct manipulation of these structures is very difficult [13]. The match in length scale between the micro transducers and the structures makes this task possible. The books by Madou [20], Kovacs [21] and Gad-el-Hak [22] provide excellent sources for micro science and technology.

Current usage of MEMS related technology is very extensive and growing rapidly. Accelerometers for automobile airbags, keyless entry systems, dense arrays of micro mirrors for high-definition optical displays, scanning electron microscope tips to image single atoms, micro-heat-exchangers for cooling of electronic circuits, reactors for
separating biological cells, blood analyzers and pressure sensors for catheter tips are but a few of the current usage. Micro ducts are used in infrared detectors, diode lasers, miniature gas chromatographs, and high-frequency fluidic control systems. Micro pumps are used for ink jet printing, environmental testing, and electronic cooling. Potential medical applications for small pumps include controlled delivery and monitoring of minute amounts of medication, manufacturing of nano liters of chemicals and development of an artificial pancreas. This emerging field not only provides miniature transducers for sensing and actuation in a domain that we could not examine in the past, but also allows us to venture into a research area in which the surface effects dominate most of the phenomena [23].

1.4. Optimization

Optimization is the methodology of maximizing or minimizing a preferred cost function while satisfying the existing constraints. Nature has plenty of examples where an optimum system is sought. In metals and alloys, the atom takes positions of least energy to form unit cells. These unit cells define the crystalline structure of materials. A liquid droplet in zero gravity is a perfect sphere, which is the geometric form of least surface area for a given volume. Tall trees form ribs near the base to strengthen them in bending. The honeycomb structure is one of the most compact packaging arrangements. Like nature organizations, businesses have also strived towards excellence. The solutions to their problems have been based mostly on judgments and experience. However, increased competition and consumer demands often require that the solutions be optimum and not just feasible solutions. A small savings in a mass produced part will result in substantial savings for the corporation. In vehicles, weight minimization can influence
fuel efficiency, increased payloads, or performance. Limited material or labor resources must be utilized to maximize profit. Often, optimization of a design process saves money for a company by simply reducing the development time [24].

In a design process, an analysis (simulation) code is coupled along with an optimizer. The process generally involves using the analysis code to obtain a flow solution for a given configuration and then trying to improve the configuration via the optimizer, using the information available. Optimization methods may be classified as gradient-based or non-gradient based. Gradient-based methods are much more popular, though the requirement for computing gradient information can often result in prohibitive costs. Initially, optimization was applied using a black-box approach, e.g., by employing finite differences to obtain gradient information. This meant that obtaining gradients involved evaluating the flow solutions for several perturbations of the design configuration. The cost of obtaining these solutions was a major factor in determining the viability of these methods. The major disadvantage of the gradient-based methods is that they tend to find the local maxima. Non-gradient based methods such as simulated annealing, genetic algorithms and response surface methods have superiority in escaping from the local minimums, however, it should be noted that these methods require evaluation of a large number of flow solutions, and, in general, may not be feasible except in parallel computing environments.

For the historical development of the optimization methods, the followings are listed in many textbooks related to the optimization such as [24]. The use of a gradient method (requiring derivatives of the functions) for minimization was first presented by Cauchy in 1847. Modern optimization methods were pioneered by Courant [1943],
Dantzig [1951], and Karush Kuhn Tucker who derived the KKT optimality conditions for constrained problems. Thereafter, particularly in the 60s, several numerical methods to solve nonlinear optimization problems were developed. Methods for unconstrained minimization include the conjugate gradient methods of Fletcher-Reeves [1954] and the variable metric methods of Davidon-Fletcher-Powell [1959, 1963]. Constrained optimization methods were pioneered by Rosen's gradient projection method [1960], Zoutendijk's method of feasible directions [1960], the generalized reduced gradient method by Abadie, Carpenter, and Hensgen [1966]. Multivariable optimization needed efficient methods for a single-variable search. Traditional interval search methods, using Fibonacci numbers and golden section ratio were followed by the efficient hybrid polynomial-interval methods of Brent [1971]. Sequential Quadratic programming (SQP) methods for constrained minimization were developed in the 1970s. The SQP method was first published by Pshenichny in 1970 in Russian and later in a book by Pshenichny and Danilin in 1978. This method has received a lot of attention in recent years owing to its superior rate of convergence [24].

In the 1960’s, side by side with the developments in gradient-based methods, there were also developments in non-gradient methods. Most recent among these methods are genetic algorithms (Holland [25] Goldberg [26]), simulated annealing algorithms that originated from Metropolis [27] and Response Surface Methods [28]. These non-gradient based methods are gaining popularity because they alleviate the problems associated with noisy simulation data that often cause convergence difficulties for gradient-based optimization. There have also been attempts to use hybrid techniques that try to use gradient and non-gradient based methods in conjunction with each other to
improve efficiency [29]. The use of parallel processing has also been incorporated to speed up the process of aerodynamic simulation ([30], [31]) and there have been attempts to incorporate parallel computing in the design process [32].

1.5. \textbf{Shape Optimization}

As a sub branch of optimization, the shape optimization aims to minimize and maximize, through variation of a domain boundary or portion thereof, an objective function subject to constraints imposed by an/a initial/boundary value problem as well as the other physical and geometrical conditions. It arises in a variety of engineering applications. The shape optimization works by conducting many numerical flow simulations and gradually improving the shape based on the simulation results. Thus, it is a step of a fundamental importance to use an efficient and accurate flow solver.

Most of the existing literature relevant to the shape optimization of fluid systems is in the context of aerodynamics ([33], [34], [35], [36]). Shape optimization differs from the other optimization problems in one important aspect- the design space is composed of flow boundary shapes. The bigger the family of physically realistic shapes that can be replicated, the more successful the optimization can be. Ideally, the shape representation should incorporate as much geometric flexibility as possible with as few design variables as possible. This philosophy allows access to a larger design space at minimum computational cost. Currently, the standard methods of curves and surface representation are based on a finite set of control points in conjunction with parametric expressions. Techniques have matured for shape parameterization including Bezier and B-Spline methods [37]. There are numerous shape optimization studies using Bezier parameterization in the literature ([33], [35], [36], [38], [39], [40]).

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
1.6. Synthetic Jet Actuators

Among all the MEMS actuators, the synthetic jets have received a great deal of attention as a potential method for active flow control. The synthetic jet actuators show good promise as an enabling technology for innovative flow control applied to both mini- and micro-flows. Significant interest has been growing in the aerospace community in the field of flow control in recent years. In particular, an international workshop, CFDVAL2004 [41], was held in March 2004 at NASA Langley Research Center to assess state-of-the-art technology for measuring and computing aerodynamic flows in the presence of the synthetic jets.

A synthetic jet is a zero-net-mass-flow fluidic device. Because they use the same working fluid of the flow system, and the effect is based on a zero mass transfer, they do not need dedicated air/fluid supply systems, as do other blowing/suction devices, thus making them suitable for a large class of applications. Even though the jet is formed without any mass injection to the system, the net effect over one cycle of the membrane motion is a finite streamwise momentum.

In a synthetic jet, the actuating flow is generated at the orifice of a cavity by oscillating a membrane opposite to the orifice (Figure 1.1). A pair of vortices forms (a vortex ring in the 3-Dimensional case) at the orifice during the expulsion stage of the membrane oscillation cycle of the synthetic jet actuator. Fluctuating jet flow then interacts with the main flow and transfers linear momentum to the main flow.
The membrane of the synthetic jet is driven by a moving surface either by a piston or by a vibrating membrane such as piezoelectric, electrostatic, or electromagnetic actuators. When the membrane moves upward, the fluid inside the cavity is expelled through the orifice creating a vortex pair that moves downward of the flow domain by its own momentum. If the vortex structure is strong enough, the vortex is shed from the orifice by its own induced velocity. As the membrane moves downward, the same amount of fluid is entrained from the external flow into the cavity. If the vortex is strong enough, this suction stage will not affect the shed vortex. Instead, most of the entrained fluid is sucked from upstream of the flow domain. Thus, the total effect of the synthetic jet will be an increase of the momentum that is transferred to the external flow with zero net mass flux.

In summary, the oscillatory jet adds momentum to the boundary layer in two ways. During the suction part of the cycle, it draws the low momentum fluid in the boundary layer inside the cavity, thereby bringing the lower momentum fluid at the
boundary layer edge near the control surface. On the other hand, during the blowing part of the cycle, it adds the same fluid with higher momentum to the flow. The average ‘effective’ momentum added over the entire cycle replenishes the momentum deficit in the boundary layer. Therefore, the actuator not only produces momentum itself but also enhances the ability of the boundary layer to overcome adverse pressure gradients through the mixing of the low momentum fluid near the surface with the high momentum external flow.

1.7. Literature Review

Numerous studies have been done both computationally and experimentally. Although, some of these studies came up with the new applications and new characteristics of the synthetic jet, some studies used the same application in different flow configurations. The previous studies related to the synthetic jet are grouped into four parts. In the first part, fundamental studies about the synthetic jets will be given. These studies are the ones that attempt to understand the basic characteristics of the synthetic jets. In the second part, some of the applications in which the synthetic jets are successfully used as a flow control actuator will be presented. In the modeling part, recent studies on the different modeling techniques and attempts on lower modeling of synthetic jet related flows will be reviewed. Finally, in the last part, the optimization studies of the synthetic jets will be given.

1.7.1. Characterization

Perhaps the most influential work on synthetic jets was performed at Georgia Tech by Glezer et al. Their work was the first to characterize the basic performance of
the synthetic jets and their ability to affect the flow over aerodynamic surfaces. Several papers written by this group experimentally characterize the small-scale effects of the synthetic jets ([42], [43], [44]). This group has employed several methods of experimentally measuring the flow field including phase-locked Schlieren imaging, Hotwire anemometry, and smoke visualization.

In addition to characterizing the performance of a single synthetic jet, Smith and Glezer investigate the performance of two adjacent synthetic jets [43]. Interestingly, they noted that by phasing the timing of the jet actuation the direction of the resulting jet could be modified.

In their experimental study, Wu and Breuer [45] considered the application of the synthetic jet actuators to a near wall turbulent boundary layer control. When the primary objective of one application requires the actuator to generate high momentum flux, it is important that the synthetic jet actuator should operate at high Reynolds numbers and low Strouhal numbers. However, in their experiment, the actuator was operated in a very narrow range, characterized by a low Reynolds number and a high Strouhal number of the jet in order to control the near wall boundary layer phenomena. They found that, in this particular range, the actuator changed from its reversible source-sink operation to a directed jet.

Smith et al. [46] studied experimentally the synthetic jet at large Reynolds numbers and compared it with the continuous jets. They stated that the synthetic jets are wider, slower and have more momentum than similar continuous jets. They also tried to determine the necessary non-dimensional stroke length necessary for jet formation. They also found that in the far-field, a synthetic jet bears much resemblance to a continuous jet.
However, in the near-field, a synthetic jet entrains more fluid and thus grows faster than a continuous jet. The effect of $Re_{uj}$ is seen in the turbulent transition of the flow exiting the nozzle, the transition of the vortex pair, and the turbulent characteristics of the developed jet flow.

Another experimental investigation into characterizing the synthetic jets was performed by Lachowicz et al. [47] at the NASA Langley Research Center. Here, the authors attempted to theoretically and experimentally determine the scaling of a cantilever-style synthetic jet and the importance of various geometric factors such as the narrow and wide gap width, the cantilever width, and the actuation frequency. To aid in the characterization, the authors used a variety of experimental techniques including Laser-Sheet Flow Visualization and Laser Velocimetry. Both methods produced excellent results, enabling the authors to distinguish between flow types for different geometry actuators. From this information, the authors were able to determine that the synthetic jet generates three primary flow types: wall jet, free jet, and vortex flow. The type of the flow is dependent on many factors including the actuation amplitude, the Reynolds number, and gap spacing.

Cater et al. [48] attempted to measure instantaneous two-dimensional in-plane velocity fields in a plane containing the orifice axis. They used these velocity fields to investigate the existence of a self-preserving velocity profile in the far field of the synthetic jet. They also compared the mean flow quantities and turbulent statistics of the synthetic jets with measurements for 'equivalent' continuous jets in the same apparatus.

Watson et al. [49] studied experimentally the flow interactions between a pair of synthetic jets. They showed that there was a minimum spacing needed between actuators...
in order to produce a single coherent jet from each actuator in the array. Furthermore, they observed from the cross flow conditions that the combined effects of the yaw angle and the orifice spacing could either reduce or increase the amount of the coherent vorticity present in the flow. Therefore, they concluded that a desired phase differencing for flow control strategies could be achieved by altering these two parameters.

Schaeffler [50] investigated the flowfield resulting from the interaction of a synthetic jet actuator and a quiescent environment. He also examined the interaction of a synthetic jet actuator and a turbulent, flat-plate, zero pressure gradient boundary layer via stereo Particle Image Velocimetry.

A comprehensive numerical study on characterizing the synthetic jets has been performed by Rizzetta et al. at the Air Force Research Lab [51]. In this study, the authors used CFD to model a synthetic jet under a variety of conditions. The authors performed both two- and three-dimensional studies on a long, thin slotted membrane style synthetic jet. The geometry of the jet is similar to ones tested by Glezer's group at Georgia Tech. To achieve an accurate overall picture, this study first considered the flow field within the jet cavity. These results were then used as the jet boundary conditions to solve the external flow field problem. Several 2-D cases were investigated and found that both the cavity height and the Reynolds number are important parameters for the performance of the jet. The 3-D simulation detected the presence of spanwise instabilities that lead to the breakup of the coherent vortex structures. These instabilities were not present in the 2-D simulations.

Utturkar et al. [52] studied the sensitivity of the synthetic jets to the design of the jet cavity using numerical simulations. In their study, they mainly focused on examining
the effect of changes in the cavity aspect ratio and the placement of piezoelectric membrane on the flow produced by the jet. They studied the synthetic jet in a quiescent environment and in the vicinity of an external flow. The results of their study showed that the details of the cavity and the placement of the membrane did not play a crucial role in the performance of the synthetic jet in terms of the vortex dynamics and the jet velocity profile.

In another study, Utturkar et al. [53] proposed and validated a jet formation criterion for the synthetic jet actuators both experimentally and computationally. However, their criterion is only valid for relatively thick orifice plates with thickness-to-width ratios greater than approximately two.

Lee and Goldstein performed another numerical simulation [54]. In this effort, the authors attempted to develop a relatively simple methodology to quickly investigate a wide range of geometric and flow parameters. The method is based on a direct numerical simulation. Unfortunately, this simulation is only valid for very low Reynolds numbers, limiting the effectiveness of the results. The authors were able to investigate several important parameters such as the thickness of the orifice and the orifice geometry. For the range studied, the authors noted that an increased orifice thickness changes the profile of the jet but not the peak velocities. The authors also noted that the orifice geometry (sharp, cusp-shaped, and rounded) had very little effect on the resulting jet.

Mittal et al. [55] studied numerically the interaction of a synthetic jet with a flat plate boundary layer using an incompressible Navier–Stokes solver. They modeled the membrane in a realistic manner as a moving boundary in an effort to accurately compute the flow inside the jet cavity. They also investigated the so-called virtual aero–shaping
effect of the synthetic jet. They found that large mean recirculation bubbles are formed in the external boundary layer only if the jet velocity is significantly higher than the cross flow velocity.

Ravi et al. [56] examined the formation and evolution of 3D synthetic jets using direct numerical simulations. They primarily focused on examining the formation of 3D synthetic jets and the effect of changes in the jet aspect ratio on the flow produced by the jet both in quiescent and cross-flow cases.

Kiddy et al. [57] investigated the different potential synthetic jet geometries after providing a detailed overview. Kral et al. [58] performed incompressible, two dimensional simulations of a synthetic jet discharging into a quiescent environment. They investigated both laminar and turbulent synthetic jets. In their study, the flow within the cavity was not calculated and an analytic velocity profile was assumed at the orifice exit to simulate an isolated synthetic jet.

Timchenko et al. [59] studied the effect of the synthetic jet actuators including the compressibility in the numerical simulations. They modeled the membrane as a sinusoidal moving boundary. The flow had a high enough $Kn$ number value so that the continuum approach using conventional conservation equations was valid. They performed both incompressible and compressible computations.

McCormick [60] used a new type of synthetic jet actuator, the so-called “directed synthetic jet,” with a throat curved in the downstream tangential direction. They investigated the directed synthetic jets for boundary layer separation numerically. They noted that on the in-stroke of the throat velocity, vertical momentum is imparted to the flow causing the neck to ingest the approaching low momentum fluid. However, on the
out-stroke of the throat velocity, the fluid particles are re-accelerated and injected with positive axial momentum into the wall region of the boundary layer. By doing so, the ability of the boundary layer to resist separation is increased.

1.7.2. Modeling

In addition to numerical simulations, several attempts have been made to analytically model the performance of a synthetic jet. Rathnasingham and Breuer proposed one of the first synthetic jet models [61]. This model consists of two parts. The first part consists of an inviscid model of the cavity with an assumed constant velocity jet being produced at the orifice. This model also assumes that as the fluid compresses and expands, due to the oscillating piston, the resulting pressure difference drives the jet through the orifice.

Gallas et al. [62] modeled the synthetic jets as elements of an equivalent electrical circuit using conjugate power variables (Lumped Element Modeling). They analyzed the synthetic jets by means of the circuit to provide a physical insight into the dependence of the device behavior on geometry and material properties. Their results indicated that the linear composite plate theory is accurate when the clamped boundary condition is achieved. Similarly, the cavity acoustic compliance model was validated. However, as they concluded, the details of the flow in the orifice requires further study due to the acoustic mass and the resonance at the orifice.

Przekwas et al. [63] from CFD Research Corp. presented two complementary synthetic jet-modeling techniques: a three-dimensional high-fidelity model for a detailed simulation of a single jet, and a reduced "single cell" model of a jet for simulation of large arrays of synthetic jets. The high-fidelity model is also used to calibrate the...
compact model. They achieved substantial computational savings when compact models instead of high-fidelity models were used.

Redionitis et al. [64] attempted to derive and to evaluate model order reduction methods based on proper orthogonal decomposition that are suitable for the synthetic jet actuators. They used this reduced order model to investigate stable feedback control laws for the synthetic jets. The authors showed that accurate modeling of such flows in the open loop response is possible with as few as four modes.

Yamalev et al. [65] proposed and validated a new reduced-order model of a synthetic jet actuator. They used a low dimensional model of 2-D/3-D synthetic jet actuators that is based on the quasi-1-D Euler equations. The oscillating membrane was simulated as a quasi-1-D moving boundary. The authors stated that the reduced model satisfies the conservation laws and is efficient in terms of computational time. However, their model provides high accuracy if the Reynolds number based on the actuator slot width is larger than 500 and the interface between the quasi-1-D Euler equations and the 2-D Navier-Stokes equations is located more than two times slot width away from the jet exit.

Pes et al. [66] tried a different way to model the synthetic jet related flows. The authors modeled 2-D synthetic jet unsteadiness with deterministic source terms trained by a neural network. They performed simulations to characterize the behavior of an isolated jet at different frequency and amplitude values of piston oscillations for several free-stream Mach numbers. They explored detailed unsteady flow physics of the synthetic jet via instantaneous and time averaged momentum coefficient, momentum thickness, displacement thickness, and shape factor. They found that the neural network based
source terms accurately reproduce the time-averaged physics of the synthetic jet, resulting in considerable improvement in computational efficiency.

Lockerby and Carpenter [67] modeled an alternative mode of the synthetic jet actuator for use in active turbulence control. The membrane was modeled using classical thin-plate theory, with the stiffness of the attached piezo-device incorporated and clamped at its edges. The fluid motion in the cavity was not modeled for numerical economy. At the center point of the membrane, symmetry condition was enforced.

1.7.3. Applications

As previously mentioned, Georgia Tech has been very active in designing and testing of synthetic jets actuators. In addition to characterizing the small-scale flow field produced by the jets, this group has produced several groundbreaking efforts in demonstrating the effectiveness of the synthetic jets in effecting the flow around large-scale structures.

In later efforts, Glezer’s group was able to demonstrate the effect of spanwise synthetic jets on the aerodynamics of a cylinder in a free stream flow ([44], [68]). In the first investigation [44], they used smoke visualization to show the effect of the synthetic jets as the cylinder is rotated and the jet location is varied along the circumference of the cylinder. Concurrent lift and drag measurements showed a substantial increase in the lift coefficient and a similar decrease in the measured drag. The second paper [68] expands upon the previous results by including Hotwire anemometry measurements of the downstream wake and a periodic amplitude modulated control signal to investigate the time-varying nature of the jet actuation. Again, significant improvements in both lift and drag were noted with a corresponding shift in the downstream wake. For the amplitude
modulated tests, it was found that the unsteady oscillations resulting from the changing control input persist for much longer periods of time after the actuation is turned off compared to the oscillations produced by turning on the synthetic jets.

A paper by Smith et al. [69] investigates the effect of a synthetic jet array on a thick, turbulent boundary layer. In particular, the synthetic jets were ejected into a high-pressure jet exhausted from a rectangular duct over a flat plate. The authors studied two cases, one where the array of jets is located perpendicular to the flow and a second with the jet array aligned along the flow direction. Preliminary results using a single axis Hotwire sensor indicated that the second configuration, with the array aligned along the axis of the flow, is less susceptible to separation and promotes greater mixing with the boundary layer.

Fujisawa et al. [70] also investigated the possible drag reduction and the corresponding variation of the flow field around a circular cylinder. They experimentally tried to control flow over a circular cylinder in a wind tunnel with acoustic excitation, which is supplied internally through a slit to the flow over the cylinder. Their results indicate that the drag is reduced about 30% in comparison to a stationary cylinder, when the control parameters are optimized, such as the slit angle, the forcing Strouhal number, and the excitation amplitude. The corresponding flow field around the cylinder with and without control was measured by Particle Image Velocimetry.

Chen et al. [71] at NASA Langley, tested the mild maneuvering capability of the synthetic jets in a subsonic wind tunnel using a 2-D NACA 0015 airfoil model in order to assess the applicability of unsteady suction and blowing to alter the aerodynamic shape of an airfoil with a purpose to enhance lift and/or to reduce drag. They stated that the effect

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
of virtual shape change is indicated by a localized increase of surface pressure in the neighborhood of synthetic jet actuation, which causes a negative lift to the airfoil with an upper surface actuation. When they applied the actuation near the airfoil leading edge, they concluded that the stagnation line is shifted, inducing an effect similar to that caused by a small angle of attack to produce an overall lift change.

Mohseni [72] proposed compact synthetic jet actuators for low speed maneuvering, ducking, and station keeping of small underwater vehicles. He also investigated the optimization of the synthetic jets for maximal thrust generation. He used synthetic jet actuators for underwater maneuvering and propulsion in two ways: (i) to improve the low speed maneuvering and station keeping capabilities of traditional propeller driven underwater vehicles, (ii) and as a synthetic jet for flow control and drag reduction at higher cruising speeds.

Mahalingam et al. [73] studied the operation of a synthetic jet based ejector and its utility for electronic cooling at relatively low flow rates. They conducted experiments and measured the flow using Particle Image Velocimetry (PIV). They used one of the active heat sink configurations for high-power microprocessors that were developed by Mahalingam and Glezer [74]. In their study, they discussed the effect of the channel on the induced flow rate, power dissipated, heat transfer coefficient and thermal efficiency by using the simple configuration of two-dimensional synthetic jet ejectors in a rectangular channel. They compared the effectiveness of synthetic jet ejectors to a steady flow produced by a conventional fan and found that an ejector module having a flow rate within the range 3–5 CFM dissipates 40% more power than the fan at the same flow rate.
Based on the success of experimental flow control with synthetic jet actuators, several researchers have attempted to predict the effect of synthetic jets in large-scale applications using numerical techniques.

Researchers at the Boeing Company in Mesa, Arizona are interested in reducing the blade-vortex interactions (BVI) that generate high levels of noise from helicopter rotor systems. Using the comprehensive rotorcraft analysis code, CAMRAD/JA, they have investigated several areas related to the synthetic jet actuators [75]. In this paper, Hassan et al. studied the ability of constant blowing or suction on the leading edge of an airfoil to reduce BVI.

Motivated by the promising results of the constant blowing studies, Hassan and JanakiRam [76] used a modified version of the NASA Ames ARC2D Navier-Stokes flow solver to investigate the effect of the synthetic jets on the aerodynamics of a NACA 0012 airfoil. The authors were able to investigate a number of parameters including jet velocity, actuation frequency, airfoil angle of attack, and actuator position. From these cases, the authors were able to show an increase in the lift of an airfoil with the proper selection of the oscillation frequency and amplitude. However, the resulting lift comes at a cost of increased drag. The authors also noted that the effectiveness of the synthetic jets decreases with an increasing angle of attack.

In a related effort, He and Kral [77] used the Reynolds averaged Navier Stokes equations to investigate the effect of the synthetic jets on the NACA 0012 at high angles of attack. However, this study considered much higher blowing rates (on the same order as the free stream velocity) than the previous study by Hassan and JanakiRam. The
authors showed that the separated flow becomes reattached to the airfoil, which results in a 23% increase in lift and a 35% decrease in drag.

Mautner [78] numerically studied the application of wall jets to low Reynolds number channel flows to enhance mixing using Lattice-Boltzmann equations. The author used a wide range of cavity-slot geometries along with various cavity wall driving velocities.

Bae et al. [79] from Gas Turbine Laboratory at MIT, attempted to control compressor tip clearance flows in a linear cascade using three types of fluidic actuators. These actuators are Normal Synthetic Jet (NSJ; unsteady jet normal to the mean flow with zero net mass flux), Directed Synthetic Jet (DSJ; injection roughly aligned with the mean flow), and Steady Directed Jet (SDJ), mounted on the casing wall. He compared the effectiveness of each active control technique in terms of their ability to achieve: (1) reduction of tip leakage flow rate, (2) mixing enhancement between tip leakage and core flow, and (3) increase in streamwise momentum of the flow in the end wall region. He showed in his study that the NSJ provides mixing enhancement only, or both mixing enhancement and leakage flow reduction, depending on its pitchwise location. The DSJ and SDJ actuators provide streamwise momentum enhancement with a consequent reduction of clearance-related blockage.

Kandil et al. [80] came up with a new idea that uses synchronized, alternating-angle direction, oscillatory (SAADO) synthetic jets to control flow separation of a NACA 0012 airfoil that is in post-stall conditions. The authors used the fundamental frequency of the separated flow as the membrane oscillation frequency, which is obtained by using a fast Fourier transform (FTT). They achieved very promising results by means of
controlling post-stall flow separation, increasing lift, reducing drag, and increasing lift-to-drag ratio.

Baysal et al. [81], proposed a micro synthetic jet to control a microflow past a backward facing step in a channel. Authors first computed an uncontrolled flow past a backward facing step in a channel. Then, they placed a synthetic jet actuator downstream of the step where the separation occurs. Analyzing a large number of test cases, they observed that the reattachment point of the separated flow and the flow dissipation are quite sensitive to the location and the geometry of the synthetic jet as well as the parameters of the oscillating membrane. They obtained the best flow control, defined as the largest decrease in dissipation, when they increased the actuator cavity width and the membrane oscillation amplitude simultaneously.

### 1.7.4. Synthetic Jet Optimization

Gallas et al. [82] studied the optimization of a piezoelectric-driven synthetic jet actuator based on a Lumped Element Modeling. To simplify the problem, they divided the optimization problem into two parts. First, a constrained optimization of the cavity volume and the orifice dimensions of two baseline synthetic jets, each with a given piezoelectric membrane, were conducted using two different objective functions. One objective function they tried to improve is the centerline output velocity over a broad frequency range, and the other objective function they tried to maximize was again the centerline velocity but at a prescribed resonant frequency of the device. They achieved significant improvements using both objective functions for both synthetic jets. Second, the authors optimized two baseline piezoelectric membranes using two configurations. One uses the standard inner-disc piezoceramic patch bonded to a metal shim, while the
other employs an outer piezoceramic ring. In each case, the objective is to maximize the achievable volume displacement of the membrane at the coercive electric field strength of the piezoceramic, while the natural frequency of the piezoelectric membrane is constrained to be greater than or equal to the baseline designs. Both configurations yield modest (~5%) improvements for one membrane and significant improvements for the other membrane (>50%).

Catalano et al. [83] applied a response-surface method, in combination with a LES/DNS numerical approach, to minimize the drag coefficient of the flow over a circular cylinder controlled via synthetic jet actuators. They evaluated the optimization in 2-D model problems at the Reynolds numbers of 500 and 3900. They performed 3-D simulations using optimal parameters obtained from 2-D. They achieved a successful optimization process, although the flow is shown to be quite insensitive to the controls applied, and the decrease in cost function is quite small at the Reynolds number 500. The authors stated that at the Reynolds number 3900, the drag coefficient reduction is more significant according to 2-D computations, but this result is not reproduced in 3-D with the same set of control parameters.

Beratlis and Smith [84] designed a synthetic jet actuator for the cooling of a one-dimensional VCSEL (Vertical Cavity Surface Emitting Laser) array. They carried out a numerical study using a CFD code (FLUENT) to find the optimal performance of the synthetic jet for cooling. The flow was assumed to be compressible and fully turbulent. The moving membranes of the actuator cavity were modeled as planar velocity inlets with a spatially uniform velocity normal to the membrane and a sinusoidal oscillation in time with a velocity amplitude. No-slip boundary condition was prescribed on all other
walls. They attempted to find the position and the orientation of the synthetic jet relative to the VCSEL array that maximizes the heat transfer from the surface of the array. They used the angle of the jet, horizontal distance from the center of the orifice and the vertical distance from the center of the orifice as design variables. They used two different MATLAB optimization routine: Nelder-Mead simplex method and pattern search method and found the pattern search method converge faster than the simplex method in their study. They also implemented geometrical constraints in their study. They found that for an operating frequency of 350 Hz, three local optimum configurations were found with heat transfer rate of at least 10% larger than the initial prototype configuration. The average cooling rate for their optimum configuration is 49% better than the original prototype configuration.

Baysal et al. [85] coupled the flow analysis with a design methodology to improve the effectiveness of a proposed micro synthetic jet actuator. In their study, the $Kn$ number more than 0.001 was considered which indicates that the flow is in the slip flow regime and continuum hypothesis is valid only with specific boundary conditions. For demonstration, they considered a flow past a micro backward facing step and selected the amplitude, frequency, orifice width, and cavity width as the design variables of the synthetic jet actuator. They used the enstrophy of the flow recirculation bubble as the objective function to minimize. First, they conducted a design of experiments study that identified the orifice width and the cavity width as the most effective design variables. Then, constructing a response surface method, they successfully optimized and found the improved control of the flow. Their optimization resulted in over 83% reduction of the enstrophy.
1.8. Outline

This dissertation consists of seven sections. Due to the fact that this dissertation includes four major subjects (Microflow and MEMS, Flow Control, Optimization, and Synthetic Jet), a brief introduction, and then a short history of the relating subject have been presented in this section. Because the main theme is the synthetic jets, previous experimental and computational studies about the synthetic jets have also been given in this section. The previous studies have been grouped into four parts. Synthetic jet categorization, modeling, and its application in the microflow field, flow control field and also a little number of optimization studies of the synthetic jets have been given.

Immediately following this section, theoretical and numerical background on these aforementioned four subjects will be given. First, the governing equations, computational schemes, and algorithms, for the flow simulations will be introduced together with the boundary conditions and preconditioning. Then relevant information about the flow physics inside the MEMS, their modeling, and the governing equations and boundary condition modifications will be provided. Finally, the optimization formulations, will be explained briefly.

After providing the relevant information about the computational tools used in this study in Section 3, the code validation studies will be presented in Section 4. Because of no available experimental data about micro synthetic jets, the validation is achieved by splitting as microflow validation and synthetic jet validation.

Section 5 and Section 6 document the numerical results of the synthetic jets in a quiescent environment and cross flow conditions. A detailed parametric study of the synthetic jets and their associated results will be given in these sections.
In Section 7, single variable and multi variable optimization studies will be presented. In addition, two different shape optimization studies and their results will also be provided. After the optimization results, conclusion of this dissertation will be given in Section 8.

1.9. Objectives

The primary objective of this research is to assess the effectiveness of synthetic jets as microfluidic actuators, and then develop a methodology to perform design optimization. To achieve this goal, the following steps are taken:

• Study micron sized synthetic jets and focus on the physics of microflows, which is different from the conventional flow physics.

• Conduct a detailed parametric study covering the large design space for synthetic jet actuation and design variables.

• Investigate the synthetic jets both in a quiescent environment and when placed in a cross flow that is to be controlled.

• Couple Computational Fluid Dynamic (CFD) and optimization methodology to study single- and multi- variable design optimization.

• Conduct shape optimization studies using shape parameters and Bezier polynomials, which seeks an improved synthetic jet shape for better performance.

1.10. Contribution of the Research

The contribution of this thesis is twofold. First, a detailed parametric study is presented that covers a large design space of synthetic jet actuation and design variables. With the help of this study, a designer or user should be able to make a cost-performance
analysis between each variable in order to achieve a better control and at the same time satisfy any constraints in case of existing limitations on any of the design variables. The second contribution is the optimization of micro synthetic jets seeking the configuration that would be most effective. These results should help the designer or user in selecting improved synthetic jet for microfluidic applications.
2. MATHEMATICAL MODEL

Theory and formulation of the present study is divided into three parts. In the first part, the flow analysis formulations are given. In the second, theoretical information about the microflow physics is given. In the third part, brief formulations about the optimization formulations are presented.

2.1. Flow Modeling and Analysis

One of the key elements of this study is the flow field analysis. Microflow computations as well as the design optimizations studies heavily depend on the flow field analysis. A flow field solution can be obtained using a CFD algorithm. In this study, a popular, well-tested advanced flow solver known as CFL3D is used to obtain the flow field solution. The general features of the code are described in Section 3 and in addition, governing equations and detailed formulations are given in Appendix A for the CFD algorithm. More information about the CFL3D code can also be found in (Rumsey et al. [86]; Baysal et al. [87]; Bartels et al. [88]).

2.1.1. Governing Equations

The computational code CFL3D v6.3 that is employed for the present numerical calculations uses the three dimensional, compressible, time dependent Navier-Stokes equations, which will be discussed in the Appendix A. The spatial discretization involves a semi discrete finite volume formulation. Roe’s upwind flux-difference splitting technique (Hirsch [89], [90]) was used for the convective and pressure terms. On the other hand, the shear stress and heat transfer terms are centrally differenced.
2.1.2. Time Advancement

The code is advanced in time with an implicit approximate-factorization method. The implicit derivatives are written as spatially first-order accurate, which results in block-tridiagonal inversions for each sweep. However, for the solutions that utilize FDS the block-tridiagonal inversions are usually further simplified with a diagonal algorithm (with a spectral radius scaling of the viscous terms).

Because of the method in which the left-hand side of equation (A.25) is treated for computational efficiency in steady-state simulations (approximate factorization, first-order accuracy), the second-order temporal accuracy is forfeited for unsteady computations. One method for recovering the desired accuracy is to use sub-iterations. Even though two different sub-iteration strategies have been implemented in CFL3D, the "pseudo time sub-iteration (τ-TS)" method is used in this study. This method is also often referred to as the "dual time stepping" method.

2.1.3. Spatial Discretization

For the discretization of inviscid fluxes, the spatial derivatives of the convective and pressure terms are written conservatively as a flux balance across a cell. The interface flux is determined from a state-variable interpolation and a locally one-dimensional flux model. CFL3D code is capable of Flux Limiting, Flux Vector Splitting, and Flux Difference Splitting, however only the Flux Difference Splitting method is used to split these inviscid fluxes.

For the discretization of viscous fluxes, the viscous terms that represent shear stress and heat transfer effects are discretized with second-order central differences. The
second derivatives are treated as differences across cell interfaces of the first-derivative terms.

2.2. Initial and Boundary Conditions

Since equations (A.29) and (A.30) are solved by numerical time advancement, a set of initial conditions are required to start the time integration process. In this study, initial conditions are set to be the free stream conditions. If there had been a previous run, then initial conditions are read from the restart file.

On the other hand, at every time step in the iteration process boundary conditions have to be enforced explicitly on all sides of the domain as well as the physical surfaces of any objects lying in the domain. In other words, boundary conditions must be applied at each face of the computational block.

CFL3D has two types of boundary condition representations namely, cell-center and cell-face. More details of these boundary conditions can be found in [91]. Even though CFL3D is capable of numerous boundary conditions, in our study the following boundary conditions are used:

General symmetry plane, extrapolation, inflow/outflow, inviscid surface, specified pressure ratio, viscous wall, user specified density and velocity components.

General Symmetry Plane

As the name implies, symmetry is assumed across an axis. The ghost point density values are set equal to their “mirror image” counterparts.

\[
\begin{align*}
\rho_{-1} &= \rho_1 \\
\rho_{-2} &= \rho_2
\end{align*}
\]
The pressure values are assigned in the same way. The velocity components at the ghost cells are obtained as follows: Consider ghost cells at $i=1$ face. Note that the normalized contra-variant velocity $\bar{U}$ is normal to $i=\text{constant}$ face. Let $\bar{U}_i$ be the normalized contra-variant velocity at cell center. For symmetry plane, $\bar{U}$ must have opposite signs on each side of the plane. Thus

$$u_{-i} = u_i - 2\xi_x \bar{U}_i$$
$$v_{-i} = v_i - 2\xi_y \bar{U}_i$$
$$w_{-i} = w_i - 2\xi_z \bar{U}_i$$

where $\xi_x$, $\xi_y$, $\xi_z$ are the unit normals at $i=1$ face.

**Extrapolation**

In the extrapolation boundary conditions, the flow field variables at ghost points are evaluated based on zeroth-order extrapolation from the computational domain. The extrapolated values would be:

$$\rho_{-1} = \rho_1$$
$$\rho_{-2} = \rho_1$$

The same zeroth order extrapolation is used for the boundary values of the other four flowfield variables.

**Inflow/Outflow**

In the inflow/outflow boundary conditions, the far-field boundary conditions are incorporated by using locally one-dimensional characteristic boundary conditions. The velocity normal to the boundary and the speed of sound for each cell are calculated from the locally one-dimensional Riemann invariants, that is

$$R^\pm = \bar{u} \pm \frac{2a}{\gamma - 1}$$
where

\[ \tilde{u} = \bar{U} - \frac{\xi_i}{|\nabla \xi|} \]

\[ \bar{U} = \frac{\xi_i}{|\nabla \xi|} u + \frac{\xi_j}{|\nabla \xi|} v + \frac{\xi_k}{|\nabla \xi|} w + \frac{\xi_l}{|\nabla \xi|} \]

\( R^- \) can be evaluated locally from conditions outside the computational domain and \( R^+ \) can be determined locally from inside the domain. The normal velocity and speed of sound are determined from

\[ \bar{u}_{face} = \frac{1}{2} (R^+ + R^-) \]

\[ a_{face} = \frac{\gamma - 1}{4} (R^+ - R^-) \]

The Cartesian velocities are determined by decomposing the normal and tangential velocity vectors:

\[ u_{face} = u_{ref} + \frac{\xi_z}{|\nabla \xi|} (\bar{u}_{face} - \bar{u}_{ref}) \]

\[ v_{face} = v_{ref} + \frac{\xi_y}{|\nabla \xi|} (\bar{u}_{face} - \bar{u}_{ref}) \]

\[ w_{face} = w_{ref} + \frac{\xi_x}{|\nabla \xi|} (\bar{u}_{face} - \bar{u}_{ref}) \]

For inflow \( ref \Rightarrow \infty \), for outflow \( ref \) represents the values from the cell inside the domain adjacent to the boundary. The sign of the normal velocity \( \bar{U}_{face} = \bar{u}_{face} + \frac{\xi_i}{|\nabla \xi|} \) determines whether the condition is at inflow \( \bar{U}_{face} < 0 \) or outflow \( \bar{U}_{face} > 0 \). The entropy \( \frac{P}{\rho^\gamma} \) is determined using the value from outside the domain for inflow and from.
inside the domain for outflow. The entropy and speed of sound are used to determine the
density and pressure on the boundary:

\[
\rho_{\text{face}} = \left[ \frac{(a_{\text{face}})^2}{\gamma s_{\text{face}}} \right]^{\frac{1}{\gamma-1}}
\]

\[
p_{\text{face}} = \rho_{\text{face}} (a_{\text{face}})^2
\]

**User Specified Density and Velocity Components**

In this boundary condition, the user specifies density and velocity components in
the input file. The pressure, on the other hand, is extrapolated from the interior.

**Specified Pressure Ratio**

The specified pressure ratio boundary condition is generally used as the outflow
boundary condition for internal flows. A single pressure ratio, \( \frac{p}{p_{\infty}} \), is specified on
input. This pressure ratio is used to set both two ghost point pressure boundary values.
Extrapolation from inside the computational domain is used to set the boundary values
for \( \rho, u, v, \) and \( w \).

**Viscous Surface**

The viscous surfaces are applied in the present study on the walls of the domains
to enforce the no-slip condition as well as the slip boundary conditions. Viscous
boundary conditions are of cell-face type. Two pieces of auxiliary information are
supplied on input: the wall temperature (\( \mathcal{T}_w / \mathcal{T}_{\infty} \)) and the mass flow (\( C_q \)) where

\[
C_q = (\rho u_{\text{normal}}) / (\rho u)_{\infty} \quad (C_q \text{ is zero if there is no flow through the wall}).
\]

No-slip (\( u_1 = u_2 = u_3 = 0 \)) conditions are applied at the body surfaces where the
Knudsen number based on the characteristic length corresponds to the continuum flow
regime. On the other hand, for the test cases, which has a characteristic length at micron level, slip boundary conditions are implemented on the walls and this option has to be added to the code CFL3D. The slip boundary conditions will be given in Section 2.4.3.

2.3. Preconditioning

The time evolution of a fluid flow is characterized by a number of velocity scales. For instance, in a one-dimensional Euler flow, information travels at three distinct speeds, \( u, u + a \), and \( u - a \) which are the eigenvalues of the Jacobian. These are the speed of convection, forward traveling pressure waves and backward traveling pressure waves, respectively. These characteristic speeds are important because they affect the performance of the analysis code. The number of iterations that must be performed to converge to the steady state solution is determined largely by the smallest characteristic speed. Consequently, if the largest and the smallest speeds in the system are widely spread, then the system converges slowly, steady state solutions are hard to obtained and the system of equations are called "stiff" [95].

One way to overcome this issue is to use the preconditioning techniques. The local time derivative preconditioning is a technique, which modifies the partial differential equations in a fashion that does not affect the physical model but does alter the velocity scales in the system. Thus by appropriate choice of preconditioner, the spread of the characteristic speeds can be reduced and the solution convergences increases.

For this reason, the Weiss-Smith low Mach number preconditioning [96] was implemented in the analysis code. The preconditioning is very helpful in obtaining convergence for low Mach number flows. The effect of the preconditioning is shown in
Figure 2.1. In this figure, the 2-D bump embedded in the channel is solved at Mach number 0.2. Even this Mach number the effect of the preconditioning is apparent. The analysis code could not even run below about Mach 0.2 without preconditioning [97]. However, the preconditioner is really only effective for low free stream Mach numbers. For Mach numbers above roughly 0.5 or so, the preconditioning loses its effectiveness, and can adversely affect convergence.

![Figure 2.1 Effect of preconditioning on the convergence [97]](image)

Suppose the three dimensional, compressible, time dependent Navier-Stokes equations are in the form of:

$$
\frac{\partial Q}{\partial t} + A \frac{\partial Q}{\partial \xi} + B \frac{\partial Q}{\partial \eta} + C \frac{\partial Q}{\partial \zeta} = 0
$$

(2.1)
where

\[
A = \frac{\partial (\hat{F} - \hat{F}_e)}{\partial Q}, \quad B = \frac{\partial (\hat{G} - \hat{G}_e)}{\partial Q}, \quad C = \frac{\partial (\hat{H} - \hat{H}_e)}{\partial Q} \tag{2.2}
\]

Then local preconditioning matrix \( \hat{P} \) is implemented as follows:

\[
\hat{P} \frac{\partial \dot{Q}}{\partial t} + \frac{\partial Q}{\partial \xi} A + \frac{\partial Q}{\partial \eta} B + \frac{\partial Q}{\partial \zeta} C = 0 \tag{2.3}
\]

This time-derivative preconditioning destroys the time accuracy of the system, but as long as both \( \hat{P} \) and \( \hat{P}^{-1} \) exist; the steady state solution is, at least potentially unaffected ([96][98]). This equation can be rewritten by multiplying with \( \hat{P}^{-1} \):

\[
\frac{\partial \dot{Q}}{\partial t} + \hat{P}^{-1} A \frac{\partial Q}{\partial \xi} + \hat{P}^{-1} B \frac{\partial Q}{\partial \eta} + \hat{P}^{-1} C \frac{\partial Q}{\partial \zeta} = 0 \tag{2.4}
\]

The preconditioned flux jacobians are \( \hat{P}^{-1} A \cdot \hat{P}^{-1} B \cdot \hat{P}^{-1} C \). With the suitable choice of \( \hat{P} \), the spread of the eigenvalues of the preconditioned flux Jacobians can be substantially reduced. Weiss-smith preconditioner has particularly simple form and can be written as follows:

\[
\hat{P} = \begin{bmatrix}
\varepsilon & 0 & 0 & 0 \\
0 & 1 & 0 & 0 \\
0 & 0 & 1 & 0 \\
0 & 0 & 0 & 1
\end{bmatrix}, \quad \hat{P}^{-1} = \begin{bmatrix}
1/\varepsilon & 0 & 0 & 0 \\
0 & 1 & 0 & 0 \\
0 & 0 & 1 & 0 \\
0 & 0 & 0 & 1
\end{bmatrix} \tag{2.5}
\]

For more information about preconditioning and its usage in CFL3D, refer to [96], [97] and [98].

### 2.4. Physics of Microflows

For a flow of gas formed inside or around the micro scaled devices the properties of the gas are expected to vary in the flow field. As in the case of gas flows in macro
scaled devices, these changes can be caused by the distributed type of external forcing such as electrical magnetic or gravitational. The momentum heat and chemical interactions between the gas and the surfaces of the microfluidics devices that are in contact with the gas flows can also have significant influences on the properties of the gas flow. Due to the small molecular weights, the gravitational effects can be small in micro gas flows similar to what has generally been observed in macro gas flows. The effects of heat transfer through either contact surfaces or dissipation are expected to be of equal if not greater importance in microflows. However, the difference in the length scale of the devices that the gas flows are associated with brings additional concerns [100].

Many questions have been raised when the results of experiments with micro devices could not be explained via traditional flow modeling. For example, during the development of the micro motor [15], it was found that the frictional force between the rotor and the substrate is a function of the contact area. This result departs from the traditional frictional law, which says that the frictional force is linearly proportional to the normal force, only. In the micro motor case, the surface forces between the rotor and the substrate contribute to most of the frictional force. However, the traditional frictional law describes situations with a dominating body force that do not depend on the contact area [15]. Deviations from the conventional wisdom commonly found in the micro world can be exemplified as follows:

- The pressure gradient in a long microchannel was observed to be nonconstant,
- The measured flow rate in microchannel was higher than that predicted from the conventional continuum flow model,
• Load capacities of micro bearings were diminished and electric currents necessary to move micro motors were extraordinarily high,

• The dynamic response of micro machined accelerometers operating at atmospheric conditions was observed to be over-damped.

Electrostatic forces and viscous effects due to the surrounding air or liquid become increasingly important as the devices become smaller as well. On the contrary, the inertial forces tend to be quite small.

The transport of mass, momentum, and energy through the surface are significantly influenced in surface effect dominant flow structures. The small length scale of micro devices may invalidate the continuum approximation altogether. Slip flow, compressibility, rarefaction, viscous heating, thermal creep, intermolecular forces and other unconventional effects have to be taken into account. The compressibility and rarefaction are the competing effects in microflows. For example, Fluid acceleration in a long microchannel where the entrance pressure is atmospheric and the exit conditions are near vacuum affects the pressure gradient along the channel resulting in a nonlinear pressure distribution. Curvature in pressure distribution is due to compressibility effects and it increases with increased inlet to outlet pressure ratio across the channel. The effect of the rarefaction is to reduce the curvature in the pressure distribution. The viscous heating effects are due to the work done by viscous stresses (dissipation), and they are important for microflows, especially in creating temperature gradients within the domain even for isothermal surfaces. The thermal creep (transpiration) phenomenon is a rarefaction effect. For a rarefied gas flow, it is possible to start the flow with tangential
temperature gradients along the channel surface. In such a case, the gas molecules start creeping from cold toward hot.

2.4.1. Flow Regimes

The local value of the Knudsen number in a particular flow determines the degree of rarefaction and the degree of validity of the continuum model. In the continuum regimes, the number of molecular collisions is big enough and the ordinary gas dynamics equations can be used. In the continuum regime I (limit zero of the Knudsen number, Table 2.1), the transport terms in the continuum momentum and energy equations are negligible and the Navier–Stokes equations then reduce to the inviscid Euler equations. Both heat conduction and viscous diffusion and dissipation are negligible, and the flow is then approximately isentropic (i.e., adiabatic and reversible) from the continuum viewpoint, while the equivalent molecular viewpoint is that the velocity distribution function is everywhere of the local equilibrium or Maxwellian form.

<table>
<thead>
<tr>
<th>Regimes</th>
<th>Governing Equations</th>
<th>Knudsen Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuum Regime I</td>
<td>Euler Equation</td>
<td>$Kn \to 0$ ($Re \to \infty$)</td>
</tr>
<tr>
<td>Continuum Regime II</td>
<td>NS Equation with No Slip BC</td>
<td>$Kn &lt; 10^{-3}$</td>
</tr>
<tr>
<td>Slip Flow Regime</td>
<td>NS Equation with Slip BC</td>
<td>$10^{-3} \leq Kn &lt; 10^{-1}$</td>
</tr>
<tr>
<td>Transition Regime</td>
<td>Burnet or Woods Equations</td>
<td>$10^{-1} \leq Kn &lt; 10$</td>
</tr>
<tr>
<td>Free Molecular Regime</td>
<td>Deterministic MD</td>
<td>$Kn \leq 10$</td>
</tr>
</tbody>
</table>

In the continuum regime II (Table 2.1), as the Knudsen number increases transport terms in the continuum momentum and energy equations become significant and the fluid flow is governed by the conventional Navier Stokes equation with no slip boundary condition.
In the slip flow regime slip (Table 2.1), boundary conditions seem to fail and a sub-layer on the order of one mean free path known as the Knudsen layer starts to become dominant between the bulk of the fluid and the wall surface. The flow in the Knudsen layer cannot be analyzed with Navier Stokes equations. Instead, it requires a special solution of the Boltzmann equation. However, for $Kn < 0.1$, the Knudsen layer covers less than 10% of the channel height and this layer can be neglected by extrapolating the bulk gas flow towards the wall. This results in a finite velocity slip value at the wall and the corresponding flow regime is called the slip flow regime. In this regime, the flow is governed by the Navier Stokes equations and the rarefaction effects are modeled through the partial slip at the wall using Maxwell's velocity slip and von Smoluchovski temperature jump boundary conditions [100].

In the transition regime (Table 2.1), the number of collisions between molecules and the number of collisions of the molecules with the surface are almost the same. In this regime, the constitutive laws that define the stress tensor and heat flux vector fail. It requires higher order corrections to the constitutive equation, which results in the Burnet and Wood equations. These equations are derived from the Boltzmann equation based on Chapman Enskog expansion including the second order $Kn$ terms [100].

In the free molecular regime (Table 2.1), mean free path is much larger than the characteristic length. The molecules reflected from the surface collide with other molecules only after flying across a large distance hence the velocity distribution function of the oncoming molecules is not influenced that much by the presence of the body. Of course, certain assumptions must be made about the law of reflection of molecules on the surface [101].
There are also some other flow regimes that are based on the other parameters. For example, based on the Mach number \((M = u/a)\), a measure of the compressibility, the flows can be classified as subsonic, sonic and supersonic. In addition, flows can be classified based on the Reynolds number \((\text{Re} = uL/v)\) which is measure the ratio between inertial forces and viscous forces. The different regimes can be tabulated as follows:

\[
\text{Re} \ll 1 \quad \text{Viscous effects dominate inertial effects.}
\]

\[
\text{Re} \approx 1 \quad \text{Viscous effects are comparable to the inertial effects.}
\]

\[
\text{Re} \gg 1 \quad \text{Inertial effects dominate viscous effects.}
\]

When considering these differences one is faced with the question of which model to use and which boundary condition to impose in order to obtain a solution to a problem in micro domain.

2.4.2. Flow Modeling

A fluid flow can be modeled in one of two ways: as a collection of individual interacting molecules or as a continuum in which properties are continuously defined throughout space. In the continuum model the velocity, density, pressure, etc. are defined at every point in space and time, and conservation of mass, energy and momentum lead to a set of nonlinear partial differential equations (Euler, Navier–Stokes, Burnett, etc.). Fluid modeling classification is depicted schematically in Figure 2.2.
Continuum Model

The continuum model, embodied in the Navier–Stokes equations, is applicable to numerous flow situations. The model ignores the molecular nature of gases and liquids and regards the fluid as a continuous medium describable in terms of the spatial and temporal variations of density, velocity, pressure, temperature and other macroscopic flow quantities. The continuum model is easier to handle mathematically than the alternative molecular models. Continuum models should therefore be used as long as they are applicable.

Basically, the continuum model leads to fairly accurate predictions as long as the local properties such as density and velocity can be defined as averages over elements large enough compared to the microscopic structure of the fluid but small enough in comparison to the scale of the macroscopic phenomena to permit the use of differential calculus to describe them. Additionally, the flow must not be too far from
thermodynamic equilibrium. The former condition is usually satisfied, but it is the latter that usually restricts the validity of the continuum equations. The shear stress and heat flux must be expressed in terms of lower order macroscopic quantities such as velocity and temperature, and the simplest (i.e., linear) relations are valid only when the flow is near thermodynamic equilibrium. Worse yet, the traditional no-slip-boundary condition at a solid–fluid interface breaks down even before the linear stress–strain relation becomes invalid [23].

An intrinsic length scale in dilute gases is the mean free path, which measures the average distance the gas molecules travel between collisions.

\[
\lambda = \frac{1}{\sqrt{2\pi d^2 n}}
\]

where \(n\) is the number density of molecules and \(d\) is the molecular diameter. For derivation of mean free path, see ref [101]. The ratio of the mean free path of a gas to the flow characteristic length scale is defined as the Knudsen number:

\[
Kn = \frac{\lambda}{L}
\]

The characteristic dimension \(L\) can be some overall dimension of the flow, but a more precise choice is the scale of the gradient of a macroscopic quantity, as, for example, the density

\[
L = \rho/(d\rho/dx)
\]

The continuum model is valid when the \(Kn\) number is too small. As long as the continuum assumption holds, there should be no particular problem with solving the microscale problems. Numerous codes and commercial packages are presently available. However, as this condition is violated, the flow is no longer near equilibrium and the
linear relation between stress and the rate of strain and the no-slip-velocity condition are no longer valid. Similarly, the linear relation between heat flux and temperature gradient and the no-jump temperature condition at a solid–fluid interface are no longer accurate when the Kn number is not small enough.

The continuum assumption will fail for some range of parameters—for gas flows when the Kn number is larger than 0.1 and for liquid flows when the fluid is sheared at a faster rate than twice the characteristic molecular interaction frequency. When the continuum assumption fails, the continuum approach to modeling flows must also fail and a new means of modeling flows must be used. This new modeling approach is based on the molecular models in which flow field is modeled as a collection of individual interacting molecules. These are statistical and deterministic models. Deterministic model is the molecular dynamics (MD).

**Molecular Dynamics**

The MD technique is in principle a straightforward application of Newton’s second law in which the product of mass and acceleration for each molecule comprising the flow is equated to the forces on the molecule which are computed according to a model quite often the Lennard-Johnson model. The simulation begins with a set of molecules in a region of space; each assigned a random velocity corresponding to a Boltzmann distribution at the temperature of interest. The interaction between the particles is prescribed typically in the form of a two-body potential energy, and the time evolution of the molecular positions is determined by integrating Newton’s equations of motion. Because MD is based on the most basic set of equations, it is valid in principle for any flow extent and any range of parameters. The method is straightforward in
principle but has two barriers: choosing a proper and convenient potential for particular fluid and solid combinations and the massive computer resources required to simulate a reasonable flow field extent. The first difficulty is, in fact, the most challenging one. There is no completely rational methodology by which a convenient potential can be chosen. The art of MD is to pick an appropriate potential and validate the simulation results with experiments or other analytical/computational results.

Molecular dynamics simulations are highly inefficient for dilute gases where the molecular interactions are infrequent. Clearly, molecular dynamics simulations are reserved for situations where the continuum approach or the statistical methods are inadequate to compute from the first principles important flow quantities.

**Direct Simulation of Monte Carlo (DSMC)**

DSMC is a simulation technique based on kinetic theory. DSMC tracks the motion of millions of particles each of which represents a large number of molecules. DSMC uncouples molecular motions from intermolecular collisions. The molecular motions are treated deterministically while the collisions are treated statistically. DSMC is a very efficient method to compute flows in the $0.1 < Kn < 10$, although it can be used in any dilute gas flow. DSMC consists of four steps which are repeated at each time step of the simulation, moving the particle, indexing and cross-referencing the particles, simulating the particle collisions sampling macroscopic properties of the flow [102] however, DSMC has also some limitations and drawbacks these are mainly finite cell size, finite time step, boundary conditions uncertainties and uncertainties in the physical parameters. For a detailed study, refer to [100] and [103].
Beside these models, gas flows can also be modeled using the Boltzmann equation. The equation regulates the molecule distribution $\tilde{f}$ in the phase space. The macroscopic properties of the micro gas flow can be obtained by sample averaging or taking a moment of the distribution function:

$$\overline{Q} = \int Q \tilde{f} \, dc$$

However, the Boltzmann equation is mathematically difficult to solve. Various approximated forms of the Boltzmann equation and its solution method have been proposed. A method proposed independently by Chapman and Enskog uses the Knudsen number as a parameter to seek distribution functions that depart slightly from the equilibrium distribution. The magnitude of the Knudsen number is assumed to be small. The solution sought for the Boltzmann equation is as follows:

$$\tilde{f} = f_0 (1 + Kn\phi_1 + Kn^2\phi_2 + ...)$$

where $\phi_1$ and $\phi_2$ are functions of gas density, temperature, and macroscopic velocity vector and $f_0$ is the equilibrium (Maxwellian) distribution function \[100\]

The series expansion method is an important development in gas kinetic theory. The first order expansion in $Kn$ showed that the momentum and the heat transport terms are linearly related to the rate of strain and the temperature gradient producing the same Navier Stokes equations that can be derived from the continuum assumption. The second order expansion produces nonlinear as well as high order derivative terms of the flow properties for the transport of momentum and energy. These belong to the various forms of Burnett equations.
2.4.3. Governing Equations and Boundary Conditions

The flow considered in this study mostly has the Knudsen range of $0.1 > Kn > 0.001$. In order for comparisons, the flows $Kn < 0.001$ are also considered. Both of the flows lay in the continuum regime; however, the former is in the slip flow regime. As mentioned earlier, in the slip flow regime conventional Navier-Stokes equations can be used with boundary condition modifications. The derivation of the governing equations can be found in any fluid mechanics textbook [(Fox and McDonald [104] White [105])]. The governing equations used in the slip flow regime are the Navier-Stokes equations, which are in the form of (A.1).

Boundary Conditions

In the slip flow regime, the compressible Navier-Stokes equations are solved subject to the velocity slip and temperature boundary conditions. Surface boundary condition routines of the code CFL3D are modified as follows. Classic no-slip and temperature wall boundary conditions (Viscous flow boundary conditions) are augmented to account for the slip velocity and the temperature jump conditions encountered in MEMS devices. The first-order Maxwell-Smoluchovski slip-boundary conditions (Gad-el-Hak [9]; Beskok [92] and Agarwal and Yun [93]) in cartesian coordinates are:

\begin{equation}
U_s - U_w = \frac{2 - \sigma_v}{\sigma_v} \frac{\pi}{\rho} \sqrt{\frac{8RT_w}{\gamma}} \tau_s + \frac{3Pr(\gamma - 1)}{4 \gamma \rho Pr T_w} (-q_s) \tag{2.6}
\end{equation}

\begin{equation}
T_s - T_w = \frac{2 - \sigma_f}{\sigma_f} \frac{\gamma - 1}{\gamma + 1} \frac{\pi}{\rho R} \sqrt{\frac{RT_w}{\gamma}} (-q_s) \tag{2.7}
\end{equation}

In these equations, subscript $s$ denotes the slip flow variables on the solid surface of the body, and $w$ denotes the wall. $\gamma$ is the ratio of specific heats, $\rho$ is the density and $R$
is the specific gas constant; $q_n$ and $q_t$ are the normal and tangential heat-flux components and $\tau_s$ is the shear stress component pertaining to the skin friction; $U_w$ and $T_w$ are the reference wall velocity and temperature, respectively; $Pr$ is the Prandtl number.

$$Pr = \frac{C_p \mu}{k} \tag{2.8}$$

Also, $\sigma_v$ and $\sigma_T$ are the tangential momentum and energy (thermal) accommodation coefficients that determine the effectiveness of tangential momentum and energy exchange of the molecules with the walls, respectively. Thermal accommodation coefficient is defined by,

$$\sigma_T = \frac{dE_i - dE_r}{dE_i - dE_w} \tag{2.9}$$

where $dE_i$ and $dE_r$ denote the energy fluxes of incoming and reflected molecules per unit time, respectively, and $dE_w$ represents the energy flux if all the incoming molecules had been re-emitted with the energy flux corresponding to the surface temperature $T_w$. The perfect energy exchange case corresponds to $\sigma_T = 1$. Similarly, the tangential momentum coefficient can be defined for tangential momentum exchange of gas molecules with surfaces as follows:

$$\sigma_v = \frac{\tau_i - \tau_r}{\tau_i - \tau_w} \tag{2.10}$$

Here, $\tau_i$ and $\tau_r$ show the tangential momentum of incoming and reflected molecules, respectively, and $\tau_w$ is the tangential momentum of re-emitted molecules, corresponding to that of the surface (for stationary surfaces $\tau_w = 0$). The case of $\sigma_v = 0$ is called specular reflection, where the tangential velocity of the molecules reflected from the walls is
unchanged, but the normal velocity of the molecules is reversed due to the normal momentum transfer to the wall. The case of $\sigma_y$ is called diffuse reflection. In this case, the molecules are reflected from the walls with zero average tangential velocity. Therefore, the diffuse reflection is an important case for tangential momentum exchange (and thus friction) of the gas with the walls. In the present study, full diffuse reflection is assumed both for the tangential momentum and energy exchange ($\sigma_y = \sigma_T = 1$).

The slip velocity and temperature jump boundary conditions given in equations (2.6) and (2.7) are first-order in Knudsen number. The application of boundary condition formulations requires the variation of tangential velocity in the normal direction to the wall. Also coded and used herein is a second-order extension of the Maxwell’s slip velocity boundary condition (Beskok et al. [94]):

$$U_s - U_w = \frac{2 - \sigma_y}{\sigma_y} \left[ \frac{Kn}{1 - bKn} \left( \frac{\partial U}{\partial n} \right) \right]$$

where $b$ is the general slip coefficient determined analytically in the slip and early transition flow regimes. For the present examples, the value of $b = -1$ is employed and the thermal creep term has been omitted.

Equation (2.6) was first proposed by Maxwell in 1879. The second term in equation (2.6) is associated with the thermal creep phenomenon, which can be important in causing pressure variation along the channels in the presence of tangential temperature gradients. Equation (2.7) was derived by von Smoluchovski and represents a model for temperature jump effects.

Equations (2.6), (2.7) can be written after nondimensionalization with a reference velocity and temperature as follows:
where $Ec$ is the Eckert number, $AT$ is a specified temperature difference in the domain and the capital letters indicate non-dimensional quantities. Also, $n$ and $s$ denote the outward normal unit vector and tangential unit vector, respectively.

2.5. Optimization Methodology

Optimization refer to the study of problems in which one seeks to minimize or maximize a real valued function, $\tilde{J}$, by methodically selecting the values of real variables from within an allowed real numbered set, $B$. Typically, $B$ is some subset of the Euclidean space $\mathbb{R}^n$, often specified by a set of, equality and/or inequality constraints that the members of $B$ have to satisfy. The elements of $B$ are called feasible solutions. The function $\tilde{J}$ is called objective or cost function. A feasible solution that minimizes (or maximizes, if that is the goal) the objective function is called an optimal solution.

Generally, when the feasible region or the objective function of the problem does not present convexity, there may be several local minima and maxima, where a local minimum $x^*$ is defined as a point for which there exists some $\delta > 0$ so that for all $x$ such that

$$\|x - x^*\| \leq \delta;$$
the expression
\[ \tilde{J}(x^*) \leq \tilde{J}(x) \]
holds; that is to say, on some region around \( x^* \) all of the function values are greater than or equal to the value at that point.

The necessary condition for \( x^* \) to be a local minimum (Necessary condition for optimality) is \( \nabla \tilde{J}(x^*) = 0 \). A point that satisfies this condition is called stationary point, which can be a minimum, maximum, or a saddle point.

The sufficient condition for \( x^* \) to be a strict local minimum is the positive definiteness of the Hessian together with the necessary condition, which is:
\[ \nabla^2 \tilde{J}(x^*) = 0 \]
\[ \nabla^2 \tilde{J}(x^*) \text{ is positive definite} \]

A nonlinear-constrained optimization, which is encountered in most of engineering problems, can be expressed as follows:

\[
\begin{align*}
\text{Minimize} & \quad \tilde{J}(x) \\
\text{Subject to} & \quad g_i(x) \leq 0 \quad (2.15) \\
& \quad h_j(x) = 0 \\
& \quad x^L \leq x \leq x^U
\end{align*}
\]

Here \( x \) is a column vector of \( n \) real valued design variables. \( \tilde{J} \) is the objective function, \( g_i \)'s are inequality constraints, \( h_j \)'s are equality constraints and \( x^L, \text{ and } x^U \) are the side constraints for the design variables. The Lagrangian function can be defined as:
\[
L = \tilde{J} + \sum_{i=1}^{m} \lambda_i g_i + \sum_{j=1}^{l} \mu_j h_j
\]

Optimality (Karush Kuhn Tucker –KKT) conditions are:
Optimality $\nabla L = \nabla \tilde{J} + \sum_{i=1}^{m}\mu_i \nabla g_i + \sum_{j=1}^{l}\lambda_j \nabla h_j = 0$

Non-negativity $\mu_i \geq 0$

Complementarity $\mu_i g_i = 0$

Feasibility $g_i \leq 0$

$h_j = 0$

where $\mu_i$ and $\lambda_j$ are the Lagrange multipliers associated with the inequality and equality constraints.

These KKT conditions are the necessary conditions. The sufficient condition for $x^*$ to be a strict local minimum is the positive definiteness of the Hessian matrix of the Lagrangian together with the necessary condition, which is:

$\nabla^2 L(x^*) = \nabla^2 \tilde{J}(x^*) + \sum_{i=1}^{m}\mu_i \nabla^2 g_i(x^*) + \sum_{j=1}^{l}\lambda_j \nabla^2 h_j(x^*)$ is positive definite.

There are numerous gradient and non-gradient optimization routines in the literature. However, in this study the sequential quadratic programming (SQP) technique is used to solve the optimization problems. SQP is the one of the most powerful methods among the mathematical nonlinear programming techniques [106]. For more detailed information on SQP, refer to [106] and [107]. SQP method has several attractions.

- The starting point can be infeasible,
- Gradients of only active constraints are needed,
- Equality constraints can be handled in addition to inequalities,
- The method can be proved to be converged under certain conditions.

As with all gradient methods, there are two tasks: the direction finding and the step size selection. A new and improved design point is then obtained as
The optimization code used in this study does not directly deal with the equality constraints given in equations (2.15) and (2.16). Instead, if such constraints are needed (which is seldom true for engineering design) it is only necessary to add additional inequality constraints of the form \(-g_i(x) \leq 0\). In other words, two equal and opposite inequality constraints will force the function to become an equality. Therefore, in this study, we will consider only inequality constraints, remembering that equality constraints can be converted to a set of two opposite inequality constraints.

In this method, first we create a quadratic approximation to the objective function using the Taylor series expansion of the objective function. The solution of the quadratic problem is used to determine the search direction at a given point. The quadratic problem expressed as follows:

$$
\text{Minimize} \quad \nabla J(x^k)^T d + \frac{1}{2} d^T \overline{H} d
$$

Subject to \(\nabla g_i(x^k)^T d + g_i(x^k) \leq 0\) \hspace{1cm} (2.17)

Note that the design variable for this quadratic problem is the direction vector. The matrix \(\overline{H}\) is a positive definite matrix, which is initially the identity matrix. On the subsequent iterations \(\overline{H}\) is updated to approach the Hessian of the objective function.

The overall optimization process is in the form of Figure 2.3:

There are three critical parts of the optimization process: finding a usable-feasible search direction \(d^k\), finding the step size \(\alpha^k\), and checking convergence. The finding a search direction is the crucial one. The first step in finding the search direction is to determine which constraints, if any, are active or violated. Active and violated constraints can be defined as in Figure 2.4.
Figure 2.3 Optimization process flowchart

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
where $C_2$ is a small negative number and $C_1$ is a small positive number.

If there is no active or violated constraints the Fletcher-Reeves conjugate direction method [108] is used:

$$d^k = -\nabla J(x^{k-1}) + \frac{|\nabla J(x^{k-2})|^2}{|\nabla J(x^{k-2})|^2} d^{k-1} \tag{2.18}$$

At the beginning of the optimization ($k=1$), only the first term of this equation (steepest descent direction) is used. For more information about the Fletcher-Reeves method and usage in VisualDOC, refer to [108], [109], and [110].
If there is an active constraint but no violated constraints finding the search direction is stated as a new optimization sub-problem selecting search direction as a design variable:

\[
\begin{align*}
\text{Minimize} & \quad \nabla J(x^{k-1})^T d^k \\
\text{Subject to} & \quad \nabla g_j(x^{k-1})^T d^k \leq 0 \quad j \in J_c \\
& \quad (d^k)^T d^k \leq 1 
\end{align*}
\]

(2.19)

The details of this sub optimization problem is out of the scope and given in [109].

If there are one or more violated constraints we must find a new search direction back toward the feasible region even if it is necessary to increase the objective function. To achieve this, the direction finding problem is augmented with a new parameter, \( \mathcal{R} \) and the sub optimization problem becomes to find the search direction and the artificial parameter \( \mathcal{R} \):

\[
\begin{align*}
\text{Minimize} & \quad \nabla J(x^{k-1})^T d^k - \Omega \mathcal{R} \\
\text{Subject to} & \quad \nabla g_j(x^{k-1})^T d^k + \theta_j \mathcal{R} \leq 0 \quad j \in J_c \\
& \quad (d^k)^T d^k + \mathcal{R}^2 \leq 1 
\end{align*}
\]

(2.20)

The parameter \( \Omega \) is initially chosen as 5.0; however, if the current iteration does not overcome the violation, \( \Omega \) is increased by a factor of 10, but limits it to an upper bound of about 1000, in order to avoid numerical ill-conditioning.

On the other hand \( \theta_j \), push-off factor is selected as follows:

\[
\theta_j = \left[ 1 - \frac{g_j(x^{k-1})}{C_2} \right]^2 \\
\theta_j \leq 50
\]

For more information about the selection of these parameters and usage, see [110]. After solving the approximate problem to find the search direction, then we need
to calculate the Lagrange multipliers at the optimum. The step size parameter is determined by searching in direction $d^k$ using the approximate Lagrangian function (one-dimensional minimization) in order to produce sufficient decrease in the function.

$$\text{Minimize } \tilde{J}(x) + \sum_{j=1}^{m} \tilde{\mu}_j \max[0, g_j(x)]$$  \hspace{1cm} (2.21)

where

$$\tilde{\mu}_j = |\mu_j| \quad j = 1, m \quad \text{at first iteration}$$

$$\tilde{\mu}_j^k = \max \left[ |\mu_j|, \frac{1}{2}(\tilde{\mu}_j^{k-1} + |\mu_j|) \right] \quad j = 1, m \quad \text{at sub-iterations}$$

At the end of the one-dimensional minimization, the Hessian of the Lagrangian is updated using the BFGS (Broydon-Fletcher-Goldfarb-Shanno) formula:

$$\bar{H}^k = \bar{H}^{k-1} + \frac{\chi^k (\chi^k)^T}{\delta^k} \left( \frac{(\bar{H}^{k-1})^T \delta^{k-1} (\delta^{k-1})^T}{\delta^{k-1} \bar{H}^{k-1} \delta^{k-1}} \right)$$  \hspace{1cm} (2.22)

where

$$\delta^{k-1} = x^k - x^{k-1}$$

$$\chi^k = \Theta \gamma + (1 - \Theta) \bar{H}^k \delta^k$$

$$\gamma = \nabla L^k - \nabla L^{k-1}$$

$$\Theta = \begin{cases} 1 & \text{if } \delta\gamma \geq 0.2 \delta^T \bar{H} \delta \\ 0.8 \delta^T \bar{H} \delta & \text{if } 0.2 \delta^T \bar{H} \delta < \delta \gamma \end{cases}$$  \hspace{1cm} (2.23)
3. COMPUTATIONAL TOOLS

3.1. Geometry and Meshing

The geometry and hence the grid required by the analysis code is generated by using a simple FORTRAN algorithms because of the non-complex geometry. These grids are structured 2-D grids that are clustered near the orifice exit. However, for the shape optimization studies, the grids sometimes become more complicated and it is not possible to generate a good quality grid using a simple FORTRAN algorithm. One way to increase the quality of the grid is to use Laplacian smoothing. The Laplacian smoothing is a simple algorithm that moves each vertex to the geometric center of its neighboring vertices. Although this method is often effective, it does not guarantee mesh improvement and can create inverted elements. In addition, there are several kinds of powerful commercial grid generation software; however, they are user-dependent which means at each optimization iteration grid has to be supplied externally. Externally providing a grid to the optimization algorithm is too much time consuming, complicated and non-scientific, yet an automated grid generation technique is required in order to perform the optimization efficiently. For this reason, a simple FORTRAN grid generation algorithm is developed. This algorithm provides an initial grid, which is usually very poor. Then using the Mesh Quality Improvement Toolkit (MESQUITE) [111], the quality of the mesh is improved and supplied to the analysis code to perform analysis and optimization.
Mesquite (Mesh Quality Improvement Toolkit) was developed by a group of researchers from Sandia National Laboratory (SNL) and Argonne National Laboratory (ANL) for SciDAC (Scientific Discovery through Advanced Computing) within the Terascale Simulations Tools and Technology (TSTT) Center. Mesquite is designed to provide a stand-alone, portable, comprehensive suite of mesh quality improvement algorithms. Mesquite software is based on a mathematical framework that improves the mesh quality by solving an optimization problem to guide the movement of mesh vertices. The user inputs a mesh and then the quality of each vertex in the mesh is described by a local quality metric that is a function of a subset of the mesh vertices. Other user input will include an objective function, which describes the norm or the average of the quality metric that is used to define the global mesh quality. For example, $L_{\infty}$ norm will tend to improve the worst-case local quality while a $L_2^2$ norm will improve the RMS quality of the global mesh. Once the objective function is defined, the user can select a numerical optimization scheme within the mesquite such as steepest descent, conjugate gradient or feasible Newton method. Using this optimization scheme, the mesh vertices are moved by mesquite toward the vertex positions of the optimal mesh, thus improving the quality according to the criterion defined by the local quality metric.

Mesquite is a linkable software library that applies a variety of node-movement algorithms to improve the quality and/or adapt a given mesh. Mesquite uses advanced smoothing and optimization to:

- Untangle meshes,
- Provide local size control,
- Improve angles, orthogonality, and skew,
• Increase minimum edge-lengths for increased time-steps,

• Improve mesh smoothness,

• Perform anisotropic smoothing,

• Improve surface meshes, adapt to surface curvature,

• Improve hybrid meshes (including pyramids and wedges),

• Smooth meshes with hanging nodes,

• Maintain quality of moving and/or deforming meshes,

• Perform ALE rezoning,

• Improve mesh quality on and near boundaries,

• Improve transitions across internal boundaries,

• Align meshes with vector fields, and

• R-adapt mesh to the solutions using error estimates.

Mesquite improves surface or volume meshes which are structured, unstructured, hybrid, or non-conformal. Varieties of element types are permitted. Mesquite is designed to be as efficient as possible so that large meshes can be improved. Working principles of mesquite are given in Appendix B and more details about mesquite can be found in [111], [112], and [113].

3.2. Flow Solver

CFL3D is a CFD code developed at NASA Langley Research Center for solving 2-D or 3-D flows on structured grids. CFL3D solves the time-dependent conservation law form of the Reynolds-averaged Navier-Stokes equations. The spatial discretization involves a semi-discrete finite-volume approach. Upwind-biasing is used for the
convective and pressure terms, while central differencing is used for the shear stress and heat transfer terms. Time advancement is implicit with the ability to solve steady or unsteady flows. Multigrid and mesh sequencing are available for convergence acceleration. Numerous turbulence models are provided. Multiple-block topologies are possible with the use of 1-1 blocking, patching, overlapping, and embedding. CFL3D does not contain any grid generation software. Grids must be supplied externally ([97], Rumsey et al. [86]; Baysal et al. [87]; Bartels et al. [88]). Capabilities and options of the CFL3D code can be enumerated as follows:

**Discretization and Numerical Method**

- Conservation law form of the Euler or RANS equations
- Spatial discretization is semi-discrete finite-volume approach
- Upwind-Biasing is used for the convective and pressure terms
- Solves either the steady or unsteady form of the equations
- Time advancement is implicit with dual time stepping and sub-iterations
- Approximate-Factorized (AF) numerical scheme
- Explicit block boundary conditions
- Multigrid Grid sequencing
- Moving grid and mesh deformation capability

**Block Structures**

- 1-1 blocking (preferred)
- Patching
- Overlapping
- Embedding

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
• Sliding patched zone interfaces

**Turbulence Models for RANS Computation**

• 0-equation models: Baldwin-Lomax, Baldwin-Lomax with Degani-Schiff modification

• 1-equation models: Baldwin-Barth, Spalart-Almaras, including Detached Eddy Simulation (DES)

• 2-equation models: Wilcox k-ω model, Menter’s k-ω Shear Stress Transport (SST) model, Abid k-ω model, k-ω and k-ε Explicit Algebraic Stress Models (EASM), k-enstrophy model

**Computing Modes**

• Sequential or single processor (single or multiple blocks)

• Parallel processing (Message Passing Interface -MPI)

• Complex computation, which allows computation of sensitivity derivatives due to static and dynamic variables (e.g. dCL/da)

### 3.3. Optimization Code

VisualDOC is a general-purpose optimization tool that allows the user to quickly add the design optimization capabilities to almost any analysis program. It uses a powerful, intuitive graphical user interface along with state-of-the-art optimization algorithms to setup, solve, and post-process the design. VisualDOC provides gradient, non-gradient and response surface based optimization algorithms along with design of experiments. VisualDOC can be used for any design problem since it can be directed by defining which parameters may change (design variables) and measures the design
quality (responses). VisualDOC solves the design problem by calling the optimizer to modify the design variables and then calling the program that calculates the responses. It is also possible to pre/post process the design optimization data. It allows the user to use the optimization in parallel/distributed computing. For the gradient-based optimization, the user can provide gradients to VisualDOC, or VisualDOC will calculate the gradients using finite difference methods. VisualDOC has efficient interfaces to Matlab, Excel and user supplied executables. VisualDOC provides a wide variety of DOE post-processing options, including detailed ANOVA tables, residual analysis, and various plots ([109], [110], [114]). The capabilities and options of the VisualDOC are as follows:

**Design Optimization Algorithms**

- Gradient Based Optimization
  - Constrained Optimization
    - Modified Method Of Feasible Directions (MMFD)
    - Sequential Quadratic Programming (SQP)
    - Sequential Linear Programming (SLP)
    - Sequential Unconstrained Optimization (BIGDOT)
  - Unconstrained Optimization
    - Broydon-Fletcher-Goldfarb-Shanno
    - Fletcher-Reeves
    - Sequential Unconstrained Optimization (BIGDOT)
- Non-Gradient Based Optimization
  - Genetic Algorithm
  - Particle Swarm Algorithm
3.4. Computational Resources

For the compute-intensive simulations reported in this dissertation, parallel supercomputing resources have been necessary for obtaining the solution in a reasonable time. All of the computations were performed using the hardware resources of Old Dominion University High Performance Computing Laboratory (ODU-HPC). A 96-node Cluster Environment is split into two groups: large and small parallel environments. System hardware configuration of this computing environment is as follows:

- large Parallel Environment
  - 64 compute nodes
  - 4 GB RAM per node
  - Dual 2.2 MHz 64 bit AMD Opteron processors
  - 70 GB hard drives

- small Parallel Environment
  - 28 compute nodes
  - 4 GB RAM per node
  - Dual 2.4 MHz 64 bit AMD Opteron processors
  - 36 GB hard drives

1 GB Ethernet Switches
4. VALIDATION STUDIES

In this section validation of micro synthetic jet is given. Due to the fact that there is no experimental study made on the micro synthetic jets which yields both the microflow and synthetic jet validation, it is impossible to validate our results with experiments. One way to validate our study is to split the validation part into two parts. In the first part, microflow capabilities of our code validated. In the second part, the realistic modeling of the synthetic jets however, in macro flow conditions is validated with experimental results. The later validation also consists of two parts; a synthetic jet in a quiescent environment and a synthetic jet in a cross flow.

4.1. Validation of Microflow Modeling

4.1.1. Straight Microchannel

In order to validate the microflow capability of the code, flow through a two dimensional isothermal subsonic microchannel with a ratio of the channel length-to-height of 20 (x/y) is considered. The computational grid is shown in Figure 4.1. This validation study is also performed in [115] using the same geometry and the same flow conditions. The reference Mach (M) and Reynolds (Re) numbers are 0.0725 and 1.22, respectively. The reference temperature is 273 K. This flow is laminar and can be considered to be in the slip flow regime, i.e. $Kn = 0.1$. The working fluid is the diatomic nitrogen. Inlet-to-outlet pressure ratio is $p_{in}/p_{out}=2.28$. In these flow conditions, $Kn$ number at the channel outlet is $Kn_{out}=0.2$ while at inlet it is $Kn_{in}=0.088$. This geometry has also been studied by Beskok [92] using both a DSMC solver and μFlow (spectral-
element-based continuum CFD solver), and by Agarwal and Yun [93] using both a Navier Stokes solver and a higher order fluid dynamics model of Burnett equations.

![Figure 4.1 Computational grid for a straight microchannel](image)

In the present computations, both the first order (equation (2.6)) and the second order (equation (2.11)) slip velocity boundary formulations have been employed. To establish the grid independence, the computations have been performed using two different meshes with 101X25 and 201X49 cells. Both grids have produced very similar results. Excellent agreement has been obtained with the analytical solutions available for the centerline pressure distribution through the channel [116] and the results in [92] and [93] (Figure 4.2). The effect of compressibility, even at such low speeds, has also been observed. The pressure distribution is nonlinear and slightly lower in magnitude compared to the no-slip solution. The centerline pressure distribution is not deemed sensitive to the order of accuracy of the slip wall formulation and there is at least an order of magnitude difference between the no slip and slip cases. In order to see the difference clearly, the deviation of the computed centerline pressure from the analytically obtained pressure, \( \Delta(\frac{P}{P_{out}}) \), is plotted in Figure 4.3.

\[
\Delta(\frac{P}{P_{out}}) = (\frac{P}{P_{out}})_{\text{analytical}} - (\frac{P}{P_{out}})_{\text{computational}} \tag{4.1}
\]
where $p_{out}$ is the channel exit pressure, which is used to normalize the centerline pressure values.

Figure 4.2 The centerline pressure distribution through the microchannel

Figure 4.3 Deviation of centerline pressure from the analytical solution
The computed wall slip velocity distributions in comparison to the results of [92], [93] and the analytical solution [116] are presented in Figure 4.4. The first order slip boundary condition results match exactly with the analytical results, which are based on first order approximations as well. The second order results match perfectly with the Navier Stokes and the Burnett Equation solutions of [93].

![Figure 4.4 Variation of slip velocity along the microchannel wall](image)

The results of [92] are slightly different for the first half of the channel but match the other results towards the exit of the duct. The increase in the mass flow due to the wall slip is about 13% as also predicted in [93]. With these results, the implementation of both the first and the second order slip boundary condition formulations in the CFD code; have been deemed appropriate for microflows.
4.1.2. **Micro Filter**

Another validation case considered for the present study is a flow through a micro-filter. Analysis of gas flows through micro-filters requires the consideration of three fundamental issues: rarefaction, compressibility, and geometric complexity (Ahmed et al. [117]). The rarefaction is due to the small characteristic length scales of micro-filters ($L$) that are comparable to the local mean free path ($\lambda$). Compressibility effects are important when there are large density variations in the micro fluidic system, particularly, when there are pressure and/or temperature fluctuations. In its simplest form, a micro-filter is a very short channel or sudden constriction. A schematic view and characteristic dimensions of a section of a rectangular micro-filter array are presented in Figure 4.5.

![Figure 4.5 Characteristic dimensions of the micro filter](image)

Experimental and numerical studies have shown that the flow in the micro-filters strongly depends on the opening factor $\beta$, the ratio of the hole-area ($h$) to the total filter
area (L). Considering that the filter holes repeat in a periodic fashion and the symmetry between the two periodic sides, gas flow through only one-half hole is computed by imposing the symmetry boundary conditions in the streamwise direction. In the present study, the ratio of the height to the length at the hole opening is $h/l=1.5$, with an opening factor $\beta = h/L = 0.6$, where $h=1.2$, $l=0.8$ and $L=2$ microns. The gray shaded areas correspond to the physical surfaces of the micro-filter, where fully accommodating, diffuse reflection boundary conditions are applied. The surface temperature is kept at 300 K. The reference length scale used in the definition of $Kn$ and $Re$ is the filter height $h$. Computational grid is shown in Figure 4.6. Mach numbers of the flow are 0.17 at the inlet and 0.23 at the outlet. The inflow Reynolds number is 7.51. The corresponding inlet and outlet $Kn$ numbers are 0.054 and 0.071 respectively.

![Figure 4.6 Micro filter and its computational domain](image-url)
Figure 4.7 Normalized centerline velocity variations

Figure 4.8 Normalized centerline temperature variations

The current results are compared to the computational results of Ahmed et al. [117], where the geometry differs from the present one only with its rounded corners.

The streamwise velocity and temperature values, normalized with corresponding inflow...
values along the centerline of the filter, are in good agreement with those of Ahmed et al. (Figure 4.7 and Figure 4.8). The flow which is uniform up to $x/L=3.3$, starts to develop before it reaches the filter inlet located $x/L=4.3$.

4.1.3. Micro Backward Facing Step

Flow past a micro backward facing step is computed to study the effect of slip boundary conditions on a separated flow. The outlet channel height, $h_c$, is 1.25 $\mu$m with a ratio of the channel length-to-exit height of 5.6. The entry to the channel is also simulated. The channel inlet is located at $x/h_c=0.86$ and the step height $s$ is taken as $s/h_c=0.467$ (Figure 4.9). The first and second order slip boundary conditions are employed. Simulation is performed for inlet Mach number $M=0.47$ and inlet Re number $Re=80$. The outlet $Kn$ number is 0.018. Sample results are presented for an inlet-to-outlet pressure ratio of $p_{in}/p_{out}=2.32$ and inlet temperature is 330 K. The walls are kept at a constant temperature of 300 K. The working fluid is nitrogen.

![Figure 4.9 Computational grid for backward facing step](image-url)
The computational grid employed consists of two domains. The grid before the step (segment I) is 49X17, and the domain after (segment II) step has a grid of 161X33 cells (Figure 4.9). To test the grid independency, a finer grid of 97X33 cells (segment I) and 241X65 cells (segment II) have also been used. Results from both grids were identical up to the fifth digit. The mass flow is monitored for convergence until a constant mass flow is achieved throughout the channel.

For this case, the streamwise pressure variation, normalized with the inlet dynamic head, \( q_{in}=0.5\rho_{in}U_{in}^2 \), and the streamwise velocity, normalized with the local speed of sound \( a \), at about the center of entrance \( y/h_c=0.155 \) are shown in Figure 4.11 and Figure 4.10, respectively. The results compare well with the DSMC computations of Beskok [92], where the results are for a slightly different flow condition of \( Kn=0.04 \).
4.2. Validation of Synthetic Jet Modeling

4.2.1. Synthetic Jet in a Quiescent Environment

Experimental results to validate our study are taken from the CFD validation of the synthetic jets and turbulent separation control (CFDVAL2004) [41] workshop that was held March 2004 at NASA Langley Research Center.

The first validation case is an isolated synthetic jet formed by a single membrane piezoelectric actuator exhausting into the ambient quiescent air. Multiple measurement techniques including Particle Image Velocimetry (PIV), Laser Doppler Velocimetry (LDV), and Hotwire probes (HW) were used to generate experimental data. This configuration is shown in Figure 4.12. This corresponds to the case 1 of CFDVAL2004 workshop. This configuration consists of 0.05 inch (1.27 mm) rectangular slot connected to a 2 inch cavity. The slot is enclosed by a 24 inches by 24 inches box and is in the
center of the floor. The flow through the slot alternates between outflow and inflow. It is
driven by a side mounted piezoelectric membrane on the side of a narrow cavity under
the floor. The medium is air at sea level. The experiment was performed at the NASA
Langley Research Center in a temperature-controlled room by Yao et al. [118]. The
atmospheric conditions are standard atmospheric conditions at sea level and can be given
as:

\[
\begin{align*}
\text{Pressure:} & \quad 101325 \text{ Pa} \\
\text{Temperature:} & \quad 297 \text{ K} \\
\text{Viscosity:} & \quad 18.4 \times 10^6 \text{ kg/(m-s)} \\
\text{Density:} & \quad 1.185 \text{ kg/m}^3
\end{align*}
\]

![Figure 4.12 Physical domain of synthetic jet actuator [41]](image)

Although the cavity and the membrane geometry of this actuator are highly three
dimensional in the interior, the actual slot through which the fluid emerges is a high
aspect ratio rectangular slot and is modeled as a two dimensional configuration. The
two-dimensional computational grid is shown in Figure 4.13. This is a multi block
structured grid. Since the flow domain is symmetric with respect to the $x=0$ plane, the
computations are carried out only in the half domain. The computational domain consists
of 3 blocks, 209X81 for cavity, 25X49 for neck and 121X193 for the outer domain. At the $j=j_{dim}$ boundary, the symmetry boundary condition is applied. The cavity of the actual synthetic jet is too complicated and the membrane is vertically located. In our modeling, the cavity is modeled as rectangular shape and the membrane is located horizontally at the bottom of the cavity. Membrane motion is modeled as a moving boundary. For the viscous walls, no-slip, no injection, zero pressure gradient and adiabatic wall conditions are used. At the $j=l$ boundary, extrapolation boundary condition is applied. For the upper boundary, inflow/outflow boundary conditions are applied. The free stream Mach number in the exterior quiescent region is 0.001 to stimulate incompressible flow in the compressible flow code to avoid numerical difficulties at Mach zero. The corresponding Reynolds number based on the orifice width is 22 at which flow can be modeled as laminar. Therefore, the flow is assumed to be laminar in the present computation.

Figure 4.13 Computational grid for synthetic jet actuator

The membrane frequency is fixed to 444.7 Hz as was in the experiment and membrane displacement in the experiment is given in Figure 4.14. As seen in this figure
the maximum displacement is 0.15 mm and the minimum displacement is -0.42 mm, as a result the amplitude is approximately 0.3 mm. In addition, the orifice diameter and the membrane length are 0.635 mm and 47 mm respectively. In our modeling, the same orifice length is used; however, the cavity length is taken as 20 mm. In order to recompense almost two and a half decreases in the cavity with, the amplitude is increased to 0.8 from 0.3. In this workshop, most of the researchers determined the amplitude so that the maximum velocity at the slot exit matches with the experimental results of Yao [118]. The maximum velocity of the experiment is 24.8 m/s at height y=0.1 mm for LDV and 28.3 m/s at height y=0.1224 mm for PIV. Selecting the amplitude as 0.8 yields a maximum velocity of 25.7 m/s at height y=0.118 mm which is very consistent with the experimental data.

Figure 4.14 Displacement of membrane in one cycle

As stated by Vatsa [119], one of the major difficulties identified during the CFDVAL2004 workshop was the large disparity in experimental data obtained from
different measurement techniques. These experiments were conducted over several months because of this inconsistency. Finally, Yao [118] obtained the detailed field data using PIV, Hotwire, and LDV measurement techniques. However, the Hotwire measurements depart significantly from the PIV and LDV results. They demonstrate good consistency between PIV and LDV measurement techniques. Although there exist some differences in some points, they are within reasonable limits. For this reason only the PIV and LDV data are used to compare our results. 720-time step per period is used in the temporal resolution.

CFL3D code is run to 10 cycles at which a repeatable periodic state of the flow solution is achieved. The origin of the phase for experimental and computational results was fixed by shifting the phase angle of the vertical velocity profiles such that the mid value of the velocity \((V_{\text{max}}+V_{\text{min}})/2\) is at the phase of 340. This shifting process is recommended by the workshop in order to compare the results with experiment and with each participant. The time history of the vertical velocity for a complete period from computational results is compared to the experiment in Figure 4.15 at \(x=0\) and around \(y=0.1\) mm which is the closest point to the slot exit where experimental data (both PIV and LDV) available. Considering the differences between PIV and LDV, the overall agreement of computational results with the experiment is quite good at this location. The fact that the heights at which the results are taken are not the same also contributes the differences. Computational results however, agree well with the PIV results in allover the period.
4.2.2. Synthetic Jet in Cross Flow

The second test case to compare our result is the turbulent flow over a hump model. This is the case 3 of the CFDVAL2004 workshop [41]. The hump model is mounted between two glass endplate frames and both leading edge and trailing edge are faired smoothly with a wind tunnel splitter plate. This is a nominally two-dimensional experiment, although there are sidewall effects (3-D flow) near the end plates (Figure 4.16). The wall mounted hump has a chord length of \( c=0.42 \) m a maximum height of 0.0537 m and a span of 0.5842 m. A slot was located at approximately 0.65\( c \) where the separation usually occurs. The body definition and computational grid is shown in Figure 4.17. The computational and the experimental domains for the hump and wind tunnel extended from \( x=-2.14 \) to \( x=4.0 \) in the streamwise direction and from \( y=0.0 \) to \( y=0.9090 \) in the normal direction. The physical domain was nondimensionalized by the length of the hump and was located between \( 0.0<x<1.0 \). Flow conditions for the experiment were...
M=0.1 and Re=2.23x10^6 per meter (Re_c=9.36x10^5). The atmospheric conditions varied but were essentially standard atmospheric conditions at sea level in a wind tunnel vented to the atmosphere, in a temperature-controlled room. These conditions can be given as approximately:

Pressure: 101325 Pa  
Temperature: 298 K  
Viscosity: 18.4 \times 10^{-6} \text{ kg/(m-s)}  
U_\infty = 34.6 \text{ m/s}

The grid used in this study is also provided by the workshop organizers. This is a structured 2D grid, which consists of 4-multi block. The grid's top wall shape is adjusted to approximately account for side plate blockage. Note that grid extends forward to -6.39c, which was found in preliminary CFD tests to yield a "run" long enough so that the computed boundary layer thickness approximately matches that of experiment. This forward extent is longer than the actual splitter plate length. Grid numbers are 49X217, 793X217, 121X161, 65X121 for upstream, downstream, cavity and orifice, respectively.
Boundary conditions used in the simulations include adiabatic and no slip walls for the cavity walls, the hump, and the lower plate surface. The inviscid wall boundary condition is used for the top of the wind tunnel. The specified pressure ratio boundary condition at the outflow and inflow boundary conditions used in inflow. For the synthetic jet case, the membrane (bottom wall) is modeled as moving boundary as in synthetic jet in a quiescent environment.

First, the difference between two-turbulence models is compared to the experimental results at zero preconditioning (Figure 4.18). Overall, both turbulence models are in good agreement with the experimental results, however, SST [120] turbulence model gave better results than the SA [121] model. This is very apparent where the local minimum and the maximum values of the $C_p$ distribution. For this reason, SST model will be used for the remaining studies.
Figure 4.18 Comparison of two turbulence models with experiment

Next, the effect of preconditioning is investigated using the SST model. For this reason, the analysis is re-run with different preconditioning values. In Figure 4.19, the computed results with different preconditioning values are compared to the experimental result. For the zero preconditioning, the computed result disagrees with the experimental results at the local minimum maximum values of the $C_p$ distribution especially at the beginning of the hump and at the peak value of the $C_p$ distribution. Then the preconditioning values increased to the 0.5, 0.75, and 0.84 respectively. At these preconditioning values, even the change is very small and only noticeable at the peak point; the agreement with the experiment is improved as the preconditioning value increases until the 0.84. The analysis is also run using a value higher than 0.84. However, a value higher than 0.84 adversely affect the convergence and any convergent solution could not be obtained. Therefore, in all remaining synthetic jet validation cases, 0.84 is used as the preconditioning value. Comparison of computed surface pressure coefficient with experimental data for no flow control case is shown in Figure 4.20.
Having examined the effect of preconditioning and decided to use the SST for the turbulence model, now flow control using synthetic jet is validated with the experimental...
results. Even the computed values are improved by using correct preconditioning and SST model rather than SA model, there are still some differences in the $C_p$ distribution for the region downstream of the separation location. This mismatch was also observed by the most of the other participants of the CFD Validation Workshop [41], but the cause for this discrepancy remains unknown. Even the DNS study of Postl et al. [122] has this mismatch. When compared to the other participants, we would say our code much more agrees with the experimental data than the other participants [41]. In order for comparison, the results of Viken from NASA LARC, Duraisamy from University of Maryland and Krishnan from university of Arizona are given below.

Figure 4.21 Comparison of results with experimental data for flow control case
Figure 4.22 Results of Viken, S. from NASA LARC

Figure 4.23 Results of Duraisamy et al. from University of Maryland
Figure 4.24 Results of Krishnan et al. from University of Arizona
5. SYNTHETIC JET IN A QUISCENT ENVIRONMENT

A synthetic jet is a zero net mass flow fluidic device. Flow is generated at the orifice of a cavity by oscillating a membrane opposite to the orifice as shown in Figure 5.1. Fluctuating jet flow, produced by the oscillatory motion of the membrane, interacts with the external domain and transfers linear momentum to the external domain.

![Schematics of synthetic jet operation](image)

**Figure 5.1 Schematics of synthetic jet operation**

As presented in Figure 5.4, the computational domain corresponds to half of the actual physical domain. In this study, the synthetic jet parameters of importance are the membrane oscillation frequency \( (f) \), the membrane oscillation amplitude \( (A) \), the orifice width \( (d_o) \), the orifice height \( (h_o) \), the cavity height \( (H) \) and the cavity width \( (W) \). Here the membrane oscillation amplitude, the orifice width, orifice height, cavity height, and the cavity height are nondimensionalized with the characteristic length, \( l_c \). Moreover, the effect of the characteristic length scale is also investigated.
In this study, the membrane of the actuator is modeled in a realistic manner as a moving boundary to accurately compute the flow inside the actuator cavity. It is assumed that the movement of the membrane is only in the vertical direction and the position. The position and the shape of the membrane are formulated using the equation below.

\[ y(x,t) = \frac{1}{2} \left[ A + A \sin\left( \frac{x + W/4}{W/2} \pi \right) \right] \sin(2\pi f t) \]  

(5.1)

where \( t \) denotes the time. Modeled by equation (5.1), the membrane is clamped at its edges as shown in Figure 5.2. \( \omega_i = \omega t = 2\pi f t \) is called the phase of the synthetic jet membrane motion. The synthetic jet and its effects are monitored at four stages of one oscillation cycle. At the first stage, \( \omega_i = \frac{\pi}{2} \) and the membrane is at maximum amplitude level. This stage is called as minimum volume stage. At the second stage, \( \omega_i = \pi \) and the membrane is at zero amplitude value however moves to downward. This stage is called maximum ingestion period. At the third stage \( \omega_i = \frac{3\pi}{2} \) and the membrane is at its minimum amplitude value. This stage is called maximum volume stage. At fourth stage, \( \omega_i = 2\pi \) and the membrane is at zero amplitude value however moving to the upward. This stage is called maximum expulsion stage.

![Figure 5.2 Synthetic jet membrane and its maximum deflection](image)
Because the synthetic jet actuation has an unsteady nature, first the periodic flow solution has to be obtained. To ensure the solution is obtained from the limit cycle and not the transient solutions, the mass flow rate at the orifice exit is monitored during the computer runs. The left hand side plot in Figure 5.3 shows the mass flow rate at the orifice exit through 40 stages (10 membrane cycles). The y-axis in this plot represents the mass flow rate at the orifice exit and the x-axis represents the stage (quarter of a membrane cycle) numbers. As the membrane oscillates, the flow inside the cavity is expelled and ingested. The periodic solution is obtained after the 2nd cycle. The right hand side figure represents the total mass flow rate at each cycle. The total mass flow rate at each cycle is calculated as follows:

\[
\dot{M} = \frac{1}{2\pi} \int_{\omega_i=0}^{\omega_i=2\pi} \dot{m} \, d\omega_i
\]

where \( \omega_i \) represents the stage and \( \dot{m} \) is the mass flow rate at each stage. Since the synthetic jet starts to actuate from the rest, the first cycle is not included in this plot due to its large value. It is also possible to see the zero net mass features of the synthetic jets. The limit cycle solution (periodic solution) is obtained after 6th cycle. There is a difference between the solutions; the difference is less than 1%. Therefore, for all remaining cases as well as for the cross flow cases, the synthetic jet is run 10 cycles and the 10th cycle results are used to calculate the vorticity, the orifice jet velocity, and the vortex properties. However, the momentum flux values at the orifice exit are the time average of the momentum flux values at each cycle from 5th cycle to 10th cycle.
The quantitative metrics considered to determine the effectiveness of a synthetic jet are the momentum flux at the orifice exit, jet velocity, the non-dimensional vortex area, the non-dimensional circulation of the vortex, the unit circulation, and nondimensional maximum vorticity value. The distance traveled by the vortex or the vortex location is also used as another effectiveness metric.

The first metric, the momentum flux at the orifice exit, is calculated as follows:

$$P = \frac{1}{2\pi} \int_{\omega_i=0}^{\omega_i=2\pi} \int \rho U (U \cdot \hat{n}) dS d\omega_i$$

Here, the control surface (CS) is the area of the orifice exit, \( U \) is the velocity vector, and \( \hat{n} \) is the unit normal. As seen in this equation, the momentum flux is integrated through one cycle (from \( \omega_i = 0 \) to \( \omega_i = 2\pi \)). The momentum flux metric is a dimensional parameter whose unit is \( \text{kg.m/s}^2 \).

The second metric is the instantaneous jet velocity at the orifice exit. This metric is also a dimensional parameter and its unit is \( \text{m/s} \).
The third metric is the vortex area. Due to the very small structures at the order of 10 μm, it is better to use the dimensionless area. The vortex area is calculated as follows:

\[
S = \begin{cases} 
  0 & \xi < \tilde{C} \\
  x.y & \xi \geq \tilde{C}
\end{cases}
\]  

(5.3)

where \( S \) is the dimensionless vortex area, which is nondimensionalized by the characteristic length. \( \xi \) is the vorticity and defined as \( \xi = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \) and \( \tilde{C} \) is the vorticity cut off value. This value is taken as -0.05 for the clockwise rotating vortex and 0.05 for the counter clockwise rotating vortex empirically which is also used in the vorticity plots.

The fourth metric is the circulation of the vortex, which indicates the strength of the vortex. The circulation is defined as follows:

\[
\Gamma = \oint_C \mathbf{V} \cdot ds = \iint_S \xi dxdy = \iint_S (\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}) dxdy
\]  

(5.4)

The circulation is calculated in the area where the vorticity is greater than the vorticity cut off value. The circulation is also dimensionless metric.

The fifth metric is the unit circulation per unit vortex area, which is simply ratio of the circulation to the area of the vortex. The sixth metric is the maximum vorticity value at the current stage. All the metrics except first and the second metrics are nondimensional values and all the metrics except the first one show the instantaneous values.
Figure 5.4 Numerical grid for the synthetic jet configuration.

The external flow domain extends $25d_o$ vertically and $20d_o$ horizontally from the symmetry plane. Computational grid dimensions for the cavity, the orifice, and the outside domain are 209X81, 17X33, and 209X369, respectively. The baseline case, which will be referred in the text many times, is given in Table 5.1. As the throat of the cavity is one of the critical geometric parameter that influence the actuator operation, its diameter is selected as the characteristic length, $lc$, and chosen to be 10 μm for the baseline case.

<table>
<thead>
<tr>
<th>Definition</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency</td>
<td>$f$</td>
<td>300 kHz</td>
</tr>
<tr>
<td>Amplitude</td>
<td>$A/lc$</td>
<td>0.4</td>
</tr>
<tr>
<td>Orifice Width</td>
<td>$d_o/lc$</td>
<td>1</td>
</tr>
<tr>
<td>Orifice Height</td>
<td>$h_o/lc$</td>
<td>1</td>
</tr>
<tr>
<td>Cavity Width</td>
<td>$W/lc$</td>
<td>20</td>
</tr>
<tr>
<td>Cavity Height</td>
<td>$H/lc$</td>
<td>4</td>
</tr>
</tbody>
</table>
In Figure 5.5, four stages of one membrane oscillation cycle are shown. At the minimum volume stage \((\omega_i = \frac{\pi}{2})\), fluid inside the synthetic jet cavity is expelled from the orifice and a vortex (a vortex pair in a whole domain) is formed at the orifice. As will be shown later, this vortex formation is highly dependent on the selected design variables. Beside the aforementioned quantitative metrics, the visibility of the vortex structure is also a metric that can be used to determine the effectiveness of the synthetic jets. However, the visibility is a qualitative metric and can be seen in the vorticity plots. The darker the vortex area indicates the stronger the vortex hence the more efficient the synthetic jet.

![Figure 5.5 Vorticity contours at four stages of one oscillation cycle](image-url)

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
At this first stage, the vortex area has a magnitude of $S=2.8$. The circulation has a nondimensional value of $\Gamma=0.872$ at this stage. Because we only considered the left hand side of the symmetry plane, the vortices are counter clockwise rotating vortices in this half plane and therefore the vorticity values are positive. Another vortex parameter, the unit vortex strength has a nondimensional value of $\Gamma/S=0.312$, which describes the vortex strength in a unit area. The maximum vorticity is at the orifice as expected at this stage and it has a nondimensional value of $\xi_{\text{max}}=1.195$. The location of the maximum vorticity gives an idea about where the vortex is because the maximum vorticity is usually inside or approximately in the middle of the vortex core, except at the first stage. For this particular case (Figure 5.5), the evolution and the shedding of the vortex process have not been yet completed. Therefore, the vortex area and the vortex strength values as well as the maximum vorticity values and locations are not meaningful for this stage. The only thing we can look at is the unit vortex strength for this stage or the stages where the evolution and shedding of the vortex process is not completed.

In the flow domain, there is also another vortex just above the first one approximately at $y=13.69/c$. This vortex is the one that is expelled or shed from the same stage of one previous cycle. The area of this second vortex is a little bigger than the first one ($S=3.103$); however, the vortex strength is almost five times less than the first one as well. The unit vortex strength is much smaller ($\Gamma/S=0.065$) than the first one. Notice that this second vortex is not a circulating structure but a vorticity value.

During its second stage (the maximum ingestion), the synthetic jet ingests some of the fluid with low momentum above the orifice, as the membrane goes downward to zero amplitude. If the vortex formed at the first stage is strong enough, it is shed from the
orifice (Figure 5.5, \( \omega_i = \pi \)). If this formed vortex is not strong enough, this ingestion stage has negative effect on the vortex, because while the membrane is going down, some of the fluid slows down and turns back into the cavity. This phenomenon ingests the vortex back to the cavity or causes the vortex to dissipate quickly. At this second stage, the vortex has traveled to the \( y=8.98/c \) point in \( y \) direction. As a reminder, the frequency in this case is 300 kHz and the velocity of the vortex corresponds to \( 29.4 \text{ m/s} \). The vortex in this stage is scattered and its area is around \( S=6.132 \). Its strength is increased a little to \( \Gamma=0.96 \); however, the unit vortex strength is decreased almost 50% (\( \Gamma/S=0.157 \)). At this stage, the maximum vorticity value inside this vortex core is \( \xi_{\text{max}}=0.336 \). The second vortex is also seen in the flow domain but is very weak. This second vortex travels to the \( y=15.23/c \) point at an \( 11.55 \text{ m/s} \) speed. Its' area is decreased almost by half to \( S=1.477 \) from \( S=3.103 \). Its strength is also decreased to \( \Gamma=0.080 \) and resulting a unit strength value of \( \Gamma/S=0.054 \). The maximum strength of this weak vortex core is around \( \xi_{\text{max}}=0.06 \).

At the next stage, the synthetic jet is in the maximum volume stage and still ingests fluid inside the cavity. This stage may also negatively affect the vortex. The formed vortex is shed from the orifice and travels upward (Figure 5.5, \( \omega_i = \frac{3\pi}{2} \)). The vortex at the second stage travels from \( 8.98/c \) to \( 10.66/c \) at a speed of \( 12.6 \text{ m/s} \) in \( y \) direction. Vortex properties are read as follows: the vortex area \( S=5.302 \), the vortex strength \( \Gamma=0.571 \), the unit vortex strength, \( \Gamma/S=0.108 \) and the maximum vorticity value is \( \xi_{\text{max}}=0.202 \). As seen, all of the properties are decreasing gradually when compared to the second stage. Different from the previous stage, the second vortex completely disappears.
from the flow domain. Even though there is a vorticity value, this value is less than the \( \bar{C} \) vorticity cut-off value and no longer be called as a vortex.

At the last stage, the membrane goes upward from the zero amplitude and it turns back the fluid going inside the cavity and starts to expel some of the fluid (Figure 5.5, \( \omega_i = 2\pi \)). If we look at the vorticity plot, we can see that there is a little circulating structure at this stage. The vortex travels to the \( y = 12.04/c \) point at a speed of 10.35 m/s. The area of the vortex and the strength of the vortex are continuing to decrease to \( S = 4.348 \) and \( \Gamma = 0.354 \) respectively. We need to pay attention to the value of the unit vortex strength and the maximum vorticity value, which are \( \Gamma/S = 0.08 \) and \( \xi_{\text{max}} = 0.128 \) respectively. When we look at the second vortex in the first stage, the vortex is not a circulation structure and has \( \Gamma/S = 0.065 \) and maximum vorticity \( \xi_{\text{max}} = 0.085 \).

Since the second vortices in the flow domain at the first and the second stages are the same vortices that were previously shed and are exactly the same vortices that will be shed, then these vortices represent the same one vortex but in different stages. Therefore, the second vortex at the first and second stages corresponds to the vortex at the fifth and sixth stages respectively. Furthermore, multiple vortex interactions are never observed for the quiescent case studies. This will help us to understand the time evolution of the vortex and its properties or more scientifically, this will allow us to track a single vortex through the stages. In Figure 5.6, the time history of one vortex from formation to dissipation is shown. As is shown, one vortex survives until the sixth stage and then it is destroyed. Table 5.2 also shows the time history of a vortex and its properties. As seen in this table, all vortex properties except the area and the circulation in the second stage are decreasing as the time passes consistent with Figure 5.6.
Table 5.2 Tracking of vortex properties for quiescent environment

<table>
<thead>
<tr>
<th>Stage</th>
<th>S</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\text{max}}$</th>
<th>$\xi_{\text{loc}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.7990</td>
<td>0.8723</td>
<td>0.3117</td>
<td>1.1946</td>
<td>5.0627</td>
</tr>
<tr>
<td>2</td>
<td>6.1316</td>
<td>0.9593</td>
<td>0.1565</td>
<td>0.3356</td>
<td>8.9793</td>
</tr>
<tr>
<td>3</td>
<td>5.3015</td>
<td>0.5710</td>
<td>0.1077</td>
<td>0.2021</td>
<td>10.6597</td>
</tr>
<tr>
<td>4</td>
<td>4.3478</td>
<td>0.3538</td>
<td>0.0814</td>
<td>0.1275</td>
<td>12.0411</td>
</tr>
<tr>
<td>5</td>
<td>3.1032</td>
<td>0.2012</td>
<td>0.0649</td>
<td>0.0853</td>
<td>13.6877</td>
</tr>
<tr>
<td>6</td>
<td>1.4763</td>
<td>0.0795</td>
<td>0.0538</td>
<td>0.0609</td>
<td>15.2297</td>
</tr>
</tbody>
</table>

In Figure 5.7, the velocity profiles at four stages of one membrane oscillation cycle are shown. For the minimum volume stage ($\omega = \pi/2$), the velocity profile like a reverse-parabolic and it has a maximum velocity of 120 m/s however, for the maximum ingestion stage ($\omega = \pi$) the velocities are not zero. The compressible flow assumption requires that the velocity has to be zero at this stage. We can say that using a boundary
condition, which uses this compressible flow assumption, is not true especially for the microflows. The conclusion may also be drawn for the maximum expulsion stage \( \omega_i = 2\pi \). For the next stage \( \omega_i = \frac{3\pi}{2} \), the velocity profile has a flattened parabolic shape. But as we will see later, this velocity profile and its magnitude are very sensitive to the design variables and a little increment on these design variable change this profile and structure.

![Figure 5.7 Velocity profiles at four stages of one oscillation cycle](image)

**Figure 5.7 Velocity profiles at four stages of one oscillation cycle**

### 5.1. Characteristic Length Scale

The effect of the characteristic length scale is examined by varying it from \( l_c = 500 \) μm to the baseline value of 10 μm. Length scale is directly related to the Reynolds number and hence the Knudsen number. As we remember \( \text{Re} = \frac{U \cdot l_c}{\nu} \) and
\( Kn = \sqrt{\frac{\gamma \pi}{2} \frac{Re}{M}} \), the characteristic length study also indicates the Reynolds and Knudsen number studies. Because synthetic jet reacts differently for \( U \) and \( l_c \) variations, then it is better to study the effects of the characteristic length and effect of the velocity separate rather than only studying the Reynolds number or Knudsen number effect. The Knudsen number is a key non-dimensional parameter. Depending on the \( Kn \) number range a full continuum or a full free molecular analysis may be applicable. Length scale of 10 \( \mu m \) corresponds to the Knudsen number of 6.23x10\(^{-3}\), which indicates that the flow is in the slip flow regime. Thus, the conventional CFD codes should be used with appropriate boundary condition modifications. However, the characteristic length 500 \( \mu m \) corresponds to the Knudsen number of 1.25x10\(^{-4}\), which is in full continuum regime. While studying the effect of the characteristic length scale all other design variables are kept constant to their baseline values (Table 5.1) except for the frequency and the characteristic length. In this study, the frequency is set to the value of 1 kHz.

At the length scale of 500 \( \mu m \), a vortex is formed, shed from the orifice and travels through the upstream (Figure 5.8). In this figure, again the four stages of one oscillation cycle are shown. There are four vortices in the flow domain and all of them are circulating structures. However, these vortices are the ones that were formed and shed at the previous cycles. Since there are many vortices in the flow domain, this specific case is investigated in detail. Therefore, in the following figures these are the one vortex through four periods.
In Figure 5.9, the variation of the vortex area with respect to the stage is shown. At the beginning, the area increases between the first two stages then it decreases gradually. The reason for this initial increase is that in the first stage, the vortex at the orifice has not completed its formation process and at this stage, there are still some...
fluids coming up through the orifice because of the compressibility and the inertia of the fluid. This figure shows the area change of a vortex with time. As the time passes, the vortex transfers its momentum to its surroundings and the area as well as the strength of the vorticity decreases.

The next figure is the circulation of a vortex, or the strength of the vortex through the stages (Figure 5.10). The same increase between the first two stages also exists in this case. However, after the second stage the circulation decreases rapidly.

![Figure 5.10 Variation of circulation with time](image)

A more valuable parameter, the unit vortex strength, (or Unit Circulation) is shown in Figure 5.11. The unit vortex strength does not show an increase between the first two stages, because at the end of the first stage, the membrane is at its maximum level and it stops to provide a favorable pressure gradient. However, at the beginning of the second stage, the inertia of the fluid lets the fluid expel from the orifice even the
membrane provides an unfavorable pressure gradient. After the pressure gradient is more than the fluid’s inertia then the ingestion really starts. That is why the area and the circulation of the vortex still increases at the second stage. However, since there is no longer a favorable pressure gradient, the unit vortex strength and as well as the maximum vorticity do not increase between the first two stages. The unit vortex strength and the maximum vorticity (Figure 5.11 and Figure 5.12) also decrease rapidly with time.

![Graph showing variation of unit circulation with time](image)

**Figure 5.11 Variation of unit circulation with time**

In Figure 5.13, the distance traveled by this vortex, or the variation of vortex location with time is shown. As shown in this figure, the distance is increasing almost linearly or second order parabolic manner.
Figure 5.12 Variation of maximum vorticity with time

Figure 5.13 Distance traveled by the vortex

For the sake comparison, the vorticity plot for different characteristic lengths is shown in Figure 5.14. This figure is a detailed lookup of the vorticity plot near the orifice. For the $l_c=400 \, \mu m$, there are two vortices in the flow domain however, for
$l_c=300 \, \mu m$ there is only one vortex which is shown in the figure. However, this vortex is not a circulating structure but only a small vorticity value. As we go through the 10 $\mu m$, first, the vortex shedding phenomena and then the vortex formation disappear. As shown in the vorticity contours and in the vortex properties table (Table 5.3), decreasing the length scale results decreasing of the vortex strength and other vortex properties. For example, decreasing the length scale from 500 $\mu m$ to 400 $\mu m$, the distance traveled by the vortex decreases by 67% and decreasing length scale from 400 $\mu m$ to 300 $\mu m$, the distance traveled by the vortex decreases by 69%. At 200 $\mu m$ and at all remaining characteristic lengths, no vortex formation is observed.

![Figure 5.14 Plot of vorticity contours for three different characteristic lengths](image)

**Table 5.3 Vortex properties table for $l_c$ study**

<table>
<thead>
<tr>
<th>$l_c$ ((\mu m))</th>
<th>S</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{max}$</th>
<th>$\xi_{loc}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>1.4797</td>
<td>0.0727</td>
<td>0.0492</td>
<td>0.1159</td>
<td>8.3480*l_c</td>
</tr>
<tr>
<td>400</td>
<td>0.7617</td>
<td>0.0290</td>
<td>0.0381</td>
<td>0.0673</td>
<td>6.9473*l_c</td>
</tr>
<tr>
<td>300</td>
<td>0.2660</td>
<td>0.0063</td>
<td>0.0237</td>
<td>0.0308</td>
<td>6.3481*l_c</td>
</tr>
</tbody>
</table>
The effect of the characteristic length scale on the orifice jet velocity profile is clearly seen in Figure 5.15. This figure demonstrates the orifice jet velocity profiles at the third stage of the synthetic jet motion. As observed, there is a backflow near the orifice wall, because the vortex inside the cavity is strong enough and due to this counter-rotating vortex, some of the fluid near the orifice gains positive vertical momentum. This positively gained momentum decreases as we go further from the orifice wall towards the orifice centerline since we are going away from the vortex core. As depicted in this figure, the characteristic length scale increases the orifice jet velocity in both directions.
Another indicator is the variation of the momentum flux with respect to the characteristic length. As depicted in Figure 5.16, the momentum flux, $P$, increases rapidly with the characteristic length. It has a maximum $0.235 \ kg.m/s^2$ and minimum $0.000002 \ kg.m/s^2$ momentum flux values. Therefore, it is concluded that the characteristic length scale is an important design variable that affects the vortex formation and shedding.

5.2. Characteristic Length Scale at Constant $k_r$

In the previous section, we investigated the effect of the characteristic length on the vortex formation, vortex shedding, orifice jet velocity profiles, and the momentum fluxes well as the other vortex properties. Now, we will try to determine what happens if we increase the characteristic length while decreasing the frequency of the synthetic jet membrane keeping the reduced frequency constant. Reduced frequency is defined as
It is similar to the Strouhal number, which is most often used when analyzing the vortex formation and shedding. Here the reduced frequency \( k_r \) is kept constant at \( 1.45 \times 10^{-3} \) which is the reduced frequency of the case \( lc=500 \) μm of the characteristic length study.

The \( lc=500 \) μm case is the same as the one obtained in the characteristic length study. Four vortices were observed in the flow domain in Section 5.1. However, as mentioned before, these vortices are the vortices that were formed and shed from the orifice at previous cycles. Plot of vorticity contours is shown in Figure 5.17. As shown, when we decrease the characteristic length and increase the frequency so to maintain reduced frequency constant, the strength of the vortex decreases. This is not clearly understood visually by examining the first vortices but is clear by examining the second vortices. The second vortex, which is very apparent for the cases \( lc=500 \) μm and 300 μm, is very weak when \( lc=200 \) μm and completely disappears for the \( lc=100 \) μm case. However, if we further decrease \( lc \), the vortex becomes very weak for \( lc=100 \) μm and disappears when \( lc=50 \) μm and 10 μm. If we compare to the characteristic length study in Section 5.1 results; while there was no vortex formation observed after the characteristic length of 400 μm, in this case the formation and shedding of the vortex is clearly observed at \( lc=400 \) μm.

The conclusion that the vortex strength decreases as the characteristic length decreases is also verified by examining the vortex properties table (Table 5.4). In this table all the vortex properties namely, the vortex area, the circulation, the unit circulation, maximum vorticity, and the vortex location are decreasing as we decrease the
characteristic length consistent with the vorticity plot. As in the characteristic length study, the vortex is transferring its momentum to surroundings, and disappearing gradually as the time passes.

Figure 5.17 Plot of vorticity contours at different characteristic lengths

Table 5.4 Vortex properties table for $k_r$ study

<table>
<thead>
<tr>
<th>$l_c$ (μm)</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\text{max}}$</th>
<th>$\xi_{\text{loc}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>1.4797</td>
<td>0.0727</td>
<td>0.0492</td>
<td>0.1159</td>
<td>8.3480*lc</td>
</tr>
<tr>
<td>400</td>
<td>1.4060</td>
<td>0.0671</td>
<td>0.0477</td>
<td>0.1028</td>
<td>8.2605*lc</td>
</tr>
<tr>
<td>300</td>
<td>1.3711</td>
<td>0.0612</td>
<td>0.0447</td>
<td>0.0863</td>
<td>8.3042*lc</td>
</tr>
<tr>
<td>200</td>
<td>1.3277</td>
<td>0.0510</td>
<td>0.0384</td>
<td>0.0663</td>
<td>8.3480*lc</td>
</tr>
<tr>
<td>100</td>
<td>0.9574</td>
<td>0.0251</td>
<td>0.0262</td>
<td>0.0354</td>
<td>8.3042*lc</td>
</tr>
</tbody>
</table>
In Figure 5.18, there is a linear relationship between the momentum flux and the characteristic length, which was greater than linear for the characteristic length study in Section 5.1. This linear relationship starts with the minimum value of $P=0.005 \text{ kg.m/s}^2$ and has a maximum value of 0.235 $\text{kg.m/s}^2$.

![Figure 5.18 Variation of momentum flux at constant $k_c$](image)

### 5.3. Amplitude

Another important design variable is the oscillation amplitude of the membrane. In order to examine the effect of the amplitude all other design variables are kept constant at the baseline values (Table 5.1).

Beginning from the $A=0.2$ value, there is a very small and weak vortex formed at the orifice (Figure 5.19). Again, this figure shows the plot of the vorticity contours at the third stage. As we increase the amplitude, the vortex strength increases and its
dissipation rate decreases. This is seen in Figure 5.19. However, Table 5.5 gives more quantifiable results. Increasing the amplitude from 0.2 to 1.6 results in an increase in every vortex property. The vortex area, the circulation (vortex strength) and distance traveled by the vortices are linearly increasing. However, the unit circulation and the maximum vorticity have a second order parabolic relationship.

![Figure 5.19 Plot of vorticity contours for different amplitudes](image)

Table 5.5 Vortex properties table for amplitude study

<table>
<thead>
<tr>
<th>A/lc</th>
<th>S</th>
<th>( \Gamma )</th>
<th>( \Gamma/S )</th>
<th>( \xi_{\text{max}} )</th>
<th>( \xi_{\text{loc}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.20</td>
<td>0.58</td>
<td>0.03</td>
<td>0.05</td>
<td>0.06</td>
<td>6.91</td>
</tr>
<tr>
<td>0.40</td>
<td>3.36</td>
<td>0.28</td>
<td>0.08</td>
<td>0.14</td>
<td>9.03</td>
</tr>
<tr>
<td>0.60</td>
<td>5.30</td>
<td>0.57</td>
<td>0.11</td>
<td>0.20</td>
<td>10.66</td>
</tr>
<tr>
<td>0.80</td>
<td>6.82</td>
<td>0.86</td>
<td>0.13</td>
<td>0.26</td>
<td>11.98</td>
</tr>
<tr>
<td>1.00</td>
<td>8.16</td>
<td>1.15</td>
<td>0.14</td>
<td>0.31</td>
<td>13.18</td>
</tr>
<tr>
<td>1.20</td>
<td>9.80</td>
<td>1.46</td>
<td>0.15</td>
<td>0.35</td>
<td>14.34</td>
</tr>
<tr>
<td>1.40</td>
<td>11.60</td>
<td>1.79</td>
<td>0.15</td>
<td>0.39</td>
<td>15.44</td>
</tr>
<tr>
<td>1.60</td>
<td>13.36</td>
<td>2.13</td>
<td>0.16</td>
<td>0.43</td>
<td>16.53</td>
</tr>
</tbody>
</table>
As also expected, the orifice jet velocity and its magnitude increase as we increase the membrane oscillation amplitude (Figure 5.20). The velocity profile at low amplitudes looks like a flatten parabola, whereas for higher amplitude values, there is a tendency for the velocity profile to gain edge horns.

Figure 5.20 Effect of amplitude on velocity profiles at the orifice exit

Figure 5.21 demonstrates the effects of the amplitude on the local jet momentum rate, which has a linear relationship. For higher amplitude cases, much more fluid is ingested and in turn ejected, thus much more momentum flux variation is observed. For the amplitude the local jet momentum relationship, it has a minimum value of 0.05 kg.m/s² for $A=0.02$ and increases linearly to the 0.582 kg.m/s² for $A=1.6$. 

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
5.4. Frequency

Frequency dependency of the synthetic jet is examined by holding all other design variables constant at their baseline case values (Table 5.1) while varying the membrane oscillation frequency.

Starting with the value of 50 kHz, the frequency is varied through 500 kHz. In Figure 5.22, the vorticity contours at the third stage is shown only from 200 kHz to 500 kHz because in smaller values of the frequency no vortex formation and vortex shedding are observed at this stage. For 50 and 100 kHz, the vortex is formed at the first stage however; it disappeared in the second stage. For 150 kHz, the vortex survives until the third stage; however, it disappears in the third stage. For the case of 200 kHz, a very weak vortex appears in the flow domain located at \( y=10lc \). The vortex location at \( y/lc=10 \) corresponds to the distance traveled by the vortex being 5lc. Because these
traveled distances are frequency dependent, it is better to use the traveling speed of the vortex rather than the traveled distance. The vortex came at this location within two stages. Reminding that $l_c=10\ \mu m$ so the velocity is equal to:

Traveling velocity=$\frac{\text{distance} \times f}{2}=5 \times 10^{-5} \times 2 \times 10^5/2=5\ m/s$

![Figure 5.22 Plot of vorticity contours for different frequencies](image)

Increasing the frequency to 250 kHz makes the vortex stronger and more apparent. The increase in the vortex strength is also confirmed in the variation of the vortex properties table (Table 5.6). As seen in this table, all of the design variables except for the traveled distance increase when we increase the frequency to the 250 kHz. On the other hand, the distance traveled by the vortex decreases to 4.74 $l_c$, which was 5 $l_c$ for 200 kHz. However, since the time the vortex traveled this position is decreased, velocity also increases. Further increasing the frequency to 300 kHz makes the vortex stronger and makes it more visible. Likewise, for the 250 kHz case, all the vortex
properties except the vortex location increases. When we increased the frequency to 350 kHz, the unit circulation and the maximum vorticity increase. The circulation also increases numerically; however, there is only very small difference between the 300 and 350 kHz cases. On the other hand, the vortex area, traveled distance and the velocity decrease. This phenomenon is also correct for the 400 and 500 kHz except that the circulation which is now decreasing. Therefore, we may conclude that after 300 kHz, the frequency has a negative effect on the vortex. This conclusion is also verified by Figure 5.22 where the vortex formation becomes less visible as we increase the frequency after the 300 kHz point. Visibility can be interpreted as an indicator of the vortex strength.

<table>
<thead>
<tr>
<th>f(kHz)</th>
<th>S</th>
<th>r/s</th>
<th>m /ax</th>
<th>cmax</th>
<th>loc</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>200</td>
<td>1.715</td>
<td>0.099</td>
<td>0.058</td>
<td>0.070</td>
<td>10.042</td>
<td>5.042</td>
</tr>
<tr>
<td>250</td>
<td>3.130</td>
<td>0.230</td>
<td>0.073</td>
<td>0.106</td>
<td>9.743</td>
<td>5.929</td>
</tr>
<tr>
<td>300</td>
<td>3.359</td>
<td>0.284</td>
<td>0.085</td>
<td>0.135</td>
<td>9.026</td>
<td>6.039</td>
</tr>
<tr>
<td>350</td>
<td>3.108</td>
<td>0.286</td>
<td>0.092</td>
<td>0.155</td>
<td>8.261</td>
<td>5.706</td>
</tr>
<tr>
<td>400</td>
<td>2.655</td>
<td>0.253</td>
<td>0.095</td>
<td>0.163</td>
<td>7.584</td>
<td>5.168</td>
</tr>
<tr>
<td>500</td>
<td>1.662</td>
<td>0.158</td>
<td>0.095</td>
<td>0.159</td>
<td>6.458</td>
<td>3.644</td>
</tr>
</tbody>
</table>

This is clearly observed when we inspect the jet velocity profiles (Figure 5.23a). The magnitude of the orifice jet velocity increases as the frequency increases from 50 kHz to 300 kHz. Nevertheless, jet velocity profiles begin to decrease after a frequency value of 300 to 350 kHz. There is only a small difference in magnitude between 300 and 350 kHz, but after this range, the magnitude of the velocity decreases very rapidly (Figure 5.23b). In order to understand the tendency of the velocity profiles, a very high value of frequency, for example $f=1000$ kHz, is applied and found that the magnitude of the profile is still decreasing as with the frequency increases.
Figure 5.23 Effect of frequency on velocity profiles at the orifice exit
Figure 5.24 Variation of momentum flux with respect to frequency

Figure 5.24 demonstrates the variation of the momentum flux with respect to the frequency. The momentum flux responds to the frequency variation in a similar manner as the velocity profile. The momentum flux begins to increase from a minimum value of 0.005 \( \text{kg.m/s}^2 \) and after reaching its maximum value at 0.14 \( \text{kg.m/s}^2 \); it finally decreases to the value of 0.08 \( \text{kg.m/s}^2 \) as the frequency increases.

5.5. Orifice Width

In this section, the effects of the orifice width on the vortex formation and shedding, velocity profiles, and the momentum fluxes well as the vortex properties will be investigated. Orifice width is varied from \( d_o = 0.25 \) to \( d_o = 5 \) while all other design variables are kept constant at the baseline case values (Table 5.1).

Shown in Figure 5.25 are the vorticity contours for the cases when the orifice width is varied from \( d_o = 0.75 \) to \( d_o = 3.5 \). Very small (\( d_o = 0.25 \) and \( d_o = 0.5 \)) and very large (\( d_o = 4 \) and \( d_o = 5 \)) values of the orifice width are not included here, due to no vortex
formation or shedding is observed. A small vortex begins to form at the first stage of these cases and then it disappears. For the case of $d_o=0.75$, an apparent vortex is formed and shed from the orifice. Increasing the orifice width to $d_o=1$ (which is the baseline value) yields more clear hence stronger vortex. From the vortex properties table (Table 5.7) it is seen that all the vortex properties such as the vortex area, the circulation, the unit circulation, maximum vorticity, and the vortex location increase. This phenomenon also correct from $d_o=1$ case to $d_o=1.5$ case.

![Figure 5.25 Plot of vorticity contours for different orifice widths](image)

However, after making a maximum point at $d_o=1.5$ point, all the properties decrease. The vortex becomes weaker as the orifice widens further. This is also seen in the vorticity plot visually. When $d_o$ is 2.5, only a vortex like formation observed at the opening of the orifice. In this case, this formation is only a vorticity value not a circulating region. Finally, for $d_o=3.5$ no vortex formation is observed. Orifice width has
a similar effect on the vortex that of the frequency. Until $d_o=1.5$, increasing the orifice width has a favorable effect on vortex but after that point increasing the orifice width has an adverse effect.

<table>
<thead>
<tr>
<th>$d_o/le$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{max}$</th>
<th>$\xi_{loc}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.75</td>
<td>1.880</td>
<td>0.131</td>
<td>0.070</td>
<td>0.096</td>
<td>7.830</td>
</tr>
<tr>
<td>1</td>
<td>3.297</td>
<td>0.272</td>
<td>0.083</td>
<td>0.129</td>
<td>8.971</td>
</tr>
<tr>
<td>1.5</td>
<td>4.348</td>
<td>0.381</td>
<td>0.088</td>
<td>0.144</td>
<td>9.453</td>
</tr>
<tr>
<td>2</td>
<td>3.714</td>
<td>0.295</td>
<td>0.080</td>
<td>0.123</td>
<td>8.744</td>
</tr>
<tr>
<td>2.5</td>
<td>2.246</td>
<td>0.147</td>
<td>0.065</td>
<td>0.091</td>
<td>7.503</td>
</tr>
<tr>
<td>3</td>
<td>0.501</td>
<td>0.024</td>
<td>0.047</td>
<td>0.059</td>
<td>5.739</td>
</tr>
</tbody>
</table>

Figure 5.26a shows the plot of the orifice jet velocity profiles for $d_o=0.25$ to $d_o=1$ cases at the third stage. At the smaller values of the $d_o$, the orifice is too small and comparable with the boundary layer and thus a parabolic velocity profile is observed.
Figure 5.26 Effect of orifice width on velocity profiles at the orifice exit

Increasing the orifice width results in an increase in the jet velocity and the parabolic profile becomes flattened. Figure 5.26b shows the velocity profile from $d_0=1$ to $d_0=5$. As one can observe, the velocity profile within the boundary layer decreases at a small rate as the orifice width increases. However, the reduction in the velocity magnitude increases towards the centerline. The flattened parabolic velocity profile at $d_0=1$ starts to get edge horns with horns growing as we increase the orifice width.

Variation of the momentum flux also shows similar results (Figure 5.27). It increases with the orifice width, reaches its maximum value (0.21), and then begins to decrease (0.08). The difference is that, while jet velocity profile begins to decrease from $d_0=1$, the momentum flux decreases after $d_0=2$. This is because the variation of momentum flux includes four stages but the plot of the velocity profiles shows only the
third stage. When we compare the results with the vortex properties table it is seen that while the momentum reaches its maximum at point 2, the others reach their maximum at point 1.5.

![Figure 5.27 Variation of momentum flux with respect to orifice width](image)

5.6. Orifice Height

After investigating the orifice width effect, it is better to understand how the orifice height affects the synthetic jet actuator. The same range has been taken as the orifice width case (from 0.25 to 5). All other design variables are kept constant at their baseline case values (Table 5.1).

In Figure 5.28, the vorticity contour plot at the third stage is shown. For \( h_0 = 0.25 \) case, a clear vortex is seen in the flow domain. This is also a circulating structure. As shown in this figure, increasing the orifice height weakens the vortex and causes the vortex to get smaller. Consistent with the vorticity plot, all of the vortex properties are
decreasing almost linearly as we increase the orifice height (Table 5.8). After $h_o=2$, the vortex strength and the vortex area get almost half of the $h_o=0.25$ case and now these vortices are no longer circulating structures. For the case of $h_o=5$, it is nearly impossible to notice the vortex.

![Figure 5.28 Plot of vorticity contours for different orifice heights](image)

### Table 5.8 Vortex properties table for orifice height study

<table>
<thead>
<tr>
<th>$h_o/lc$</th>
<th>S</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{max}$</th>
<th>$\xi_{loc}$</th>
<th>Relative $\xi_{loc}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25</td>
<td>3.814</td>
<td>0.329</td>
<td>0.086</td>
<td>0.140</td>
<td>8.858</td>
<td>9.608</td>
</tr>
<tr>
<td>0.50</td>
<td>3.641</td>
<td>0.308</td>
<td>0.085</td>
<td>0.135</td>
<td>8.912</td>
<td>9.412</td>
</tr>
<tr>
<td>0.75</td>
<td>3.478</td>
<td>0.291</td>
<td>0.084</td>
<td>0.132</td>
<td>8.907</td>
<td>9.157</td>
</tr>
<tr>
<td>1.0</td>
<td>3.297</td>
<td>0.272</td>
<td>0.083</td>
<td>0.129</td>
<td>8.971</td>
<td>8.971</td>
</tr>
<tr>
<td>1.5</td>
<td>2.933</td>
<td>0.234</td>
<td>0.080</td>
<td>0.122</td>
<td>8.995</td>
<td>8.495</td>
</tr>
<tr>
<td>2.0</td>
<td>2.508</td>
<td>0.192</td>
<td>0.076</td>
<td>0.113</td>
<td>9.102</td>
<td>8.102</td>
</tr>
<tr>
<td>2.5</td>
<td>2.068</td>
<td>0.150</td>
<td>0.072</td>
<td>0.103</td>
<td>9.177</td>
<td>7.677</td>
</tr>
<tr>
<td>3.0</td>
<td>1.658</td>
<td>0.113</td>
<td>0.068</td>
<td>0.093</td>
<td>9.325</td>
<td>7.325</td>
</tr>
<tr>
<td>3.5</td>
<td>1.279</td>
<td>0.082</td>
<td>0.064</td>
<td>0.083</td>
<td>9.492</td>
<td>6.992</td>
</tr>
<tr>
<td>4.0</td>
<td>0.944</td>
<td>0.057</td>
<td>0.060</td>
<td>0.075</td>
<td>9.720</td>
<td>6.720</td>
</tr>
<tr>
<td>5.0</td>
<td>0.397</td>
<td>0.021</td>
<td>0.053</td>
<td>0.061</td>
<td>10.376</td>
<td>6.376</td>
</tr>
</tbody>
</table>
In Figure 5.29a, the orifice jet velocity profile is shown for the cases of $h_0 = 0.25$, 0.5, 0.75 and 1. The velocity profiles are very close to each other and the maximum velocities vary only 2-3 m/s, but they are increasing as the orifice height increases. After $h_0 = 1$, the orifice jet velocity decreases as the orifice height increases roughly linearly. One can state that there is a conflict between the vorticity plot and the velocity plot. In the vorticity plot, the $h_0 = 0.25$ case gives the best vorticity and the vorticity decreases simultaneously after this point. However, even in the velocity profile plot there is a little difference, the $h_0 = 1$ case looks like the best. This is because the velocity profile represents the third stage of the synthetic jet actuator. However, the vorticity plot shows the vorticity that formed in the first stage and shed in the second stage. The first stages of these cases have similar effect but in the second stage the $h_0 = 0.25$ case has a big difference (for example maximum velocities are 10 m/s for $h_0 = 1$ and 25 m/s for $h_0 = 0.25$) and this difference results in a better vortex appearance. For $h_0 = 1 - 5$ cases, the orifice jet velocity profile decreases as the orifice height increases (Figure 5.29b).

Figure 5.30 shows the variation of the momentum flux for different orifice heights. Consistent with the vorticity and the velocity profile plots, after a slight increase in the momentum, it decreases as the orifice height increases. We have to note that the maximum momentum flux 0.135 kg.m/s$^2$ is and the minimum momentum flux is 0.035 kg.m/s$^2$. 

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 5.29 Effect of orifice height on velocity profiles at the orifice exit
5.7. Cavity Height

In this section, we will investigate the effect of the cavity height on the synthetic jet actuator. Beginning with \( H=2 \), the cavity height is increased up to \( H=12 \). The other design variables are kept constant at their baseline case (Table 5.1). The vorticity contour plot is shown in Figure 5.31. In this plot, \( H=10 \) and \( H=12 \) cases are not included, because in these cases there is no vortex structure obtained in the flow domain. Although there is a vorticity value for the \( H=10 \) as seen in Table 5.9, it is so small that it cannot be seen in the vorticity plot. As can be seen, increasing the cavity height negatively affect the vortex structure. For the case of \( H=2 \), there is a clear vortex; however, in \( H=8 \) this vortex structure is so small that it is very hard to notice. Moreover, in the cases that are not shown in the plot (\( H=10 \) and \( H=12 \)) even this small vortex structure disappears.
Consistent with the vorticity plot, all of the vortex properties are decreasing as we increase the cavity height (Table 5.9).

![Figure 5.31 Plot of vorticity contours for different cavity heights](image)

<table>
<thead>
<tr>
<th>$H/l_c$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\text{max}}$</th>
<th>$\xi_{\text{loc}}$</th>
<th>Relative $\xi_{\text{loc}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>4.614</td>
<td>0.419</td>
<td>0.091</td>
<td>0.153</td>
<td>8.733</td>
<td>10.733</td>
</tr>
<tr>
<td>4</td>
<td>3.297</td>
<td>0.272</td>
<td>0.083</td>
<td>0.129</td>
<td>8.971</td>
<td>8.971</td>
</tr>
<tr>
<td>6</td>
<td>1.977</td>
<td>0.141</td>
<td>0.071</td>
<td>0.100</td>
<td>9.574</td>
<td>7.574</td>
</tr>
<tr>
<td>8</td>
<td>0.937</td>
<td>0.057</td>
<td>0.061</td>
<td>0.075</td>
<td>10.720</td>
<td>6.720</td>
</tr>
<tr>
<td>10</td>
<td>0.235</td>
<td>0.012</td>
<td>0.051</td>
<td>0.057</td>
<td>12.253</td>
<td>6.253</td>
</tr>
</tbody>
</table>

The velocity jet profile (Figure 5.32) and the variation of momentum flux plot (Figure 5.33) are consistent with the vorticity plot and the vortex properties table. The momentum flux decreases rapidly as the cavity height increases. The maximum to momentum flux is $0.18 \text{ kg.m/s}^2$ and the minimum momentum flux is about $0.02 \text{ kg.m/s}^2$. 

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 5.32 Effect of cavity height on velocity profiles at the orifice exit

Figure 5.33 Variation of momentum flux with respect to cavity height

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
5.8. Cavity Width

Sensitivity of the synthetic jet flow to the cavity width is studied by varying the cavity width from 10 to 30. The other design variables are kept constant at their baseline case (Table 5.1). For the case of the cavity width equals 10, the formed vortex quickly dissipates with the influence of ingested fluid at the third stage (Figure 5.34, W=10). At this stage, although one can see a vortex-like formation, it is not a circulating vortex. As the cavity width is widened to W=15, a clear vortex formation and shedding are observed. Widening the cavity to 20 makes the vortex stronger, which can be identified visually by checking the vorticity plot. This increase in the vorticity can also be verified by the vortex properties table (Table 5.10). Further increasing the cavity width to 22.5 and 25 does not appear to change the strength of the vorticity visually, but from the vortex properties table, the difference can be noticed. At the value of 22.5, the vortex area hits a maximum point and then it begins to decrease as the cavity width increases. The vortex strength however, hits a maximum point at W=25 and then begins to decrease as the cavity width increases. From Table 5.10, the differences between the cases for 22.5 and 25 are less than 1% for both the vortex area and the vortex strength. On the other hand, the unit circulation and the maximum vorticity value make maximum at W=27.5 and the difference between W=22.5 and W=27.5 cases is almost 1%. There is only a little bit difference in the vortex location and the circulation values when the cavity width becomes 30. For the cases tested, vortex formation and vortex shedding are observed in all except when the width is W=10.
More detailed results can be extracted from the jet velocity profiles. Figure 5.35a displays the jet velocity profiles at different cavity widths. As presented in this plot, the magnitude of the jet velocity increases as the cavity width increases from 10 to 22.5. After this point, increase in the cavity width results in a decrease of the jet velocity.
Figure 5.35 Effect of cavity width on velocity profiles at the orifice exit

magnitude, which looks like greater than linear decrease. In Figure 32a, doubling the cavity width nearly results in doubling the magnitude of the jet velocity. However, in
Figure 5.35b, widening the cavity width 22.5 to 30 only results in less than 15% decreases in the magnitude. Variation of the momentum flux also shows similar results. It has a parabolic nature, which peaks between 22.5 and 25 (Figure 5.36). It has a maximum momentum flux 0.14 $kg.m/s^2$ and minimum momentum flux of 0.07 $kg.m/s^2$.

![Figure 5.36 Variation of momentum flux with respect to cavity width](image)

**5.9. Summary**

Figure 5.37 is the summary of our results for the quiescent case. It also helps to compare the effects of all design variables on the momentum flux. As we can see, the amplitude has the largest effect on momentum flux with a linear relationship and thus it is the most influencing design variable. The characteristic length scale also has a large effect on the momentum rate. As one can expect, the synthetic jet flow is very sensitive to the characteristic length. On the other hand, the orifice width, the frequency, and the
cavity width have increasing-decreasing effects. Among these three design variables, the orifice width has the relatively biggest effect and the cavity width has the smallest effect.
6. SYNTHETIC JET IN A CROSS FLOW

The flow of a fluid through a microfluidic channel can be characterized by the Reynolds number. Due to the small dimensions of microchannels, the Re is usually much less than 100, often less than 1.0. In this Reynolds number regime, the flow is completely laminar. The transition to turbulent flow generally takes place for Reynolds numbers around or larger than 2000. Laminar flow provides a means by which molecules can be transported in a relatively predictable manner through microchannels. However, even at Reynolds numbers below 100, it is possible to have momentum-based phenomena such as flow separation. On the other hand, gaseous flows are often compressible in microdevices even at low Mach numbers. Viscous effects can cause sufficient pressure drop and density changes for the flow to be treated as compressible.

In this section, the synthetic jet is placed in a microchannel and all the analyses are carried out with the cross flow in order to investigate the effectiveness of synthetic jet as an actuator as well as the interactions between the synthetic jet and cross flow. In addition to design variables investigated for the quiescent flow cases, effects of the incoming flow velocity are studied.

Computational domain consists of three blocks (Figure 6.1). The first one is the external domain extends from -75lc to 75lc which corresponds to (715X193) computational grid. In order to reduce the boundary condition effects, the external domain was selected that long. The second and third domains are the orifice and the cavity domains, which have (33X33), and (201X65) computational grid points, respectively. For all other cases except for incoming flow velocity study, the free stream
Mach number is taken as 0.0578 which corresponds to the velocity of 20 m/s. The Reynolds number based on the orifice width is taken as 68.7. All solid walls are applied isothermal and slip velocity boundary condition applied. Inflow (specified density and velocity components) and outflow (specified pressure ratio) boundary conditions are applied at the entrance and at the exit of the microchannel. Grid blocks are split into 24 blocks in order to use the code parallel. Again, membrane is modeled as a moving boundary condition.

The periodicity of the flow solution is obtained again monitoring the mass flow rate. It was found that the synthetic jet flow solution reaches a periodic, limit cycle solution after four or five cycles. Therefore, as was in the quiescent case, the synthetic jet is run ten cycle and the tenth cycle results are used for the vorticity, the orifice jet velocity, and the vortex properties. However, the momentum flux values at the orifice are the time average of the momentum flux values at each cycle after the fifth cycle.

![Figure 6.1 Computational grid for cross flow case](image)

The baseline case, which will be referred many times in the next sections, is given in Table 6.1. Different from the quiescent case the characteristic length (orifice width) was taken 50 µm instead of 10 µm because cross flow has unfavorable effect on the synthetic jet actuator. It affects the vortex structure and hence the momentum. Although apparent vortex formation and shedding phenomena are observed for the quiescent case,
it is not possible to observe any vortex structure for the cross flow case even the same synthetic jet configuration is used. In order to overcome this negative effect, as we saw in quiescent case results, we need to increase either the amplitude or the characteristic length. The increase in the amplitude too much does not make sense since sometimes it is hard to increase physically. In order to examine the effects of the design variables, the characteristic length is selected as 50 μm. In this flow configuration it corresponds to $Kn=1.25 \times 10^{-3}$.

<table>
<thead>
<tr>
<th>Definition</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency</td>
<td>$f$</td>
<td>25 kHz</td>
</tr>
<tr>
<td>Amplitude</td>
<td>$A/lc$</td>
<td>0.4</td>
</tr>
<tr>
<td>Orifice Width</td>
<td>$d_o/lc$</td>
<td>1</td>
</tr>
<tr>
<td>Orifice Height</td>
<td>$h_o/lc$</td>
<td>1</td>
</tr>
<tr>
<td>Cavity Width</td>
<td>$W/lc$</td>
<td>20</td>
</tr>
<tr>
<td>Cavity Height</td>
<td>$H/lc$</td>
<td>4</td>
</tr>
<tr>
<td>Inflow Velocity</td>
<td>$U_o$</td>
<td>20 m/s</td>
</tr>
</tbody>
</table>

In Figure 6.2 the vorticity plots at four stages of one oscillation cycle are depicted. As explained previously, $\omega_t = \frac{\pi}{2}$ corresponds to the minimum volume stage. At this stage, the fluid inside the cavity is expelled and the membrane is at its maximum amplitude. The vortex is formed at the orifice and it is in the early stage of shedding from the orifice. This is also a circulating structure. As we look closely at the orifice, we can see that there is only one vortex, which is a clockwise rotating vortex. The other vortex, which is the pair of this one, is bursted by the incoming flow. In the cavity however, there is a vortex pair and these vortices look symmetric. At this stage, the dimensionless area of the vortex is 3.288. The other dimensionless vortex properties such as the circulation, the unit circulation, and maximum vorticity are -0.689, -0.209,
and -0.868 respectively. The negative signs in these values indicate the clockwise rotating vortex. The maximum vorticity location is at $x=0.5l_c$ which indicates that the vortex shedding has not been completed yet.

![Figure 6.2 Vorticity contours at four stages of membrane oscillation cycle](image)

In this plot, there are also two other vortices which are clockwise rotating and they are at the $x=9.9l_c$ and $x=19.1l_c$ respectively. The second vortex is the same vortex that was shed from the orifice at the same stage of one previous cycle. This second vortex is also a circulating structure. This second vortex traveled $9.9l_c$ distance in one cycle. As the time passes, the vortex transfers its momentum to surroundings and hence the vortex and its properties are decreased. For example, the area is reduced to 2.8. This reduction is also observed from the vorticity plot. The other vortex properties are also reduced to -2.31 for the circulation, -0.08 for the unit circulation and -0.127 for maximum vortex strength.
It is clearly seen that the counter clockwise rotating vortex, which is a vorticity value for this second vortex completely disappears. The third vortex in the flow domain is the one that was shed from the orifice at the same stage of two previous cycle. This third vortex traveled $9.2lc$ distance in one cycle. As we compare to the second vortex, all the vortex properties are decreased gradually to the values of 1.01 for area, -0.053 for the circulation -0.053 for the unit circulation and -0.058 for the maximum vorticity. However, this time the vortex is only a vorticity value rather than a circulating structure. The vortex properties of these vortices are shown in Table 6.2. In this table, the first vortex corresponds to the vortex at stage number 1, the second vortex corresponds to the vortex at stage number 5 and the third one corresponds to the vortex at stage number 9.

<table>
<thead>
<tr>
<th>Stage</th>
<th>S</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\text{max}}$</th>
<th>$\xi_{\text{loc}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.288</td>
<td>-0.689</td>
<td>-0.209</td>
<td>-0.868</td>
<td>0.500</td>
</tr>
<tr>
<td>2</td>
<td>4.505</td>
<td>-0.557</td>
<td>-0.124</td>
<td>-0.277</td>
<td>4.217</td>
</tr>
<tr>
<td>3</td>
<td>3.020</td>
<td>-0.352</td>
<td>-0.117</td>
<td>-0.211</td>
<td>4.683</td>
</tr>
<tr>
<td>4</td>
<td>3.015</td>
<td>-0.293</td>
<td>-0.097</td>
<td>-0.164</td>
<td>7.065</td>
</tr>
<tr>
<td>5</td>
<td>2.802</td>
<td>-0.231</td>
<td>-0.082</td>
<td>-0.127</td>
<td>9.862</td>
</tr>
<tr>
<td>6</td>
<td>2.610</td>
<td>-0.185</td>
<td>-0.071</td>
<td>-0.100</td>
<td>12.299</td>
</tr>
<tr>
<td>7</td>
<td>2.260</td>
<td>-0.142</td>
<td>-0.063</td>
<td>-0.080</td>
<td>14.235</td>
</tr>
<tr>
<td>8</td>
<td>1.807</td>
<td>-0.104</td>
<td>-0.058</td>
<td>-0.068</td>
<td>16.781</td>
</tr>
<tr>
<td>9</td>
<td>1.012</td>
<td>-0.053</td>
<td>-0.053</td>
<td>-0.058</td>
<td>19.092</td>
</tr>
<tr>
<td>10</td>
<td>0.140</td>
<td>-0.006</td>
<td>-0.045</td>
<td>-0.051</td>
<td>22.370</td>
</tr>
</tbody>
</table>

For the second stage (Figure 6.2, $\omega_t = \pi$), the membrane goes downward from the maximum amplitude to the zero point, which is called as maximum ingestion stage. As we mentioned earlier, this stage has a negative effect on the vortex because when the membrane goes downward, it creates an adverse pressure gradient and some of the fluid near the orifice slows down and turns back into the cavity. If the vortex is not strong enough and is not far enough from the orifice, the vortex will dissipate quickly because of
this suction. At this stage, the counter clockwise rotating vortex is nearly dissipated. There are three vortices in the flow domain. The first one is just above the orifice, the second one is at $x=12.3l/c$ and the third one is at $x=22.4l/c$. The first vortex is almost completed its shedding process and the vortex properties are maximum for the first vortex. However, the third vortex in this stage is the smallest and the weakest when we compared to the other vortices in all over the stages. These three vortices and their vortex properties are shown in Table 6.2. The stage numbers are 2 for the first vortex, 6 for the second vortex and the 10 for the third vortex. As shown in this table, all the vortex properties are increasing except the area for the first vortex.

There is a nearly symmetrical vortex forming at the orifice inside the cavity at the second stage. This forming vortex completes its evolution process and shed from the orifice through the membrane in the next stage (third stage) and then it combines with the vortex that is already in the cavity and amplifies it. Because of the viscosity, this inside-the-cavity vortex pair dissipates slowly as the time passes, but the vortex formed at this stage re-amplifies and this cyclic phenomenon goes on.

The next stage is the maximum volume stage, in which the membrane goes to the minimum amplitude value (Figure 6.2, $\omega = \frac{3\pi}{2}$). At these ingestion stages (second and third stages), the low momentum fluid upward to the orifice is ingested as well as that of downward. In this stage, the first vortex in the flow domain became a circular shape. This stage also has a negative effect on the vortex structure. This first vortex only traveled from 4.21 to 4.68 because of two consecutive ingestion stages. The decrease in the area and the circulation is around 33% however, this high decrease is not observed in the unit circulation and the maximum vorticity value. The second vortex in the flow
domain is at $x=14.2lc$. All of the vortex properties are still decreasing gradually as is seen in Table 6.2. In this table, the first and second vortices correspond to the stage numbers 3 and 7 respectively.

At the last stage (Figure 6.2, $\omega_t = 2\pi$), the membrane goes from minimum amplitude value to zero amplitude value. This stage is called the maximum expulsion stage. There are two vortices in the flow domain. The first one is at the distance of $x=7lc$ and the second one is at the distance of $x=16.8lc$. At the orifice, the new vortex starts to develop; however, it is too small for now. The first vortex traveled around $2.5lc$, which is almost five times bigger than the distance that the first vortex traveled in the previous stage.

Since the second and third vortices in the flow domain are the same vortices that were previously shed and are exactly the same vortices that will be shed then these vortices represent the same single vortex but in different stages. For example, the second vortex at stage one corresponds to the vortex at fifth stage and the third vortex at this stage corresponds to the same vortex at ninth stage. As was in the quiescent cases in Section 5, multiple vortex interactions are never observed for the cross flow cases. Having no multiple vortex interactions, it is possible to track and monitor a single vortex and its properties through the stages. In Figure 6.3, the time history of one vortex from formation to dissipation is shown. As shown, one vortex survives until the tenth stage and then it is destroyed. Table 6.2 also shows the time history of a vortex and its properties. As seen in this table, all vortex properties except the area in the second stage are gradually decreasing as the time passes consistent with Figure 6.3.
In Figure 6.4, the velocity distributions at the orifice for these four stages are shown. The velocity distribution at the orifice gives valuable information. As we can see later, sometimes it is impossible to predict the velocity distribution and the distribution highly depends on the design variables. In addition, we are convinced that to use a synthetic jet as a boundary condition not always yields a correct result. For example, using a parabolic or blunted parabolic velocity profile or to use an incompressible flow
like boundary condition (parabolic-zero-reverse parabolic-zero) is not correct. Another point in this plot is the slip velocities that are difficult to observe. This is due to the high jet velocities. A detailed plot will show that these slip velocities are one order of magnitude less than the jet flow.

![Figure 6.4 Velocity profiles at four stages of membrane oscillation cycle](image)

6.1. Characteristic Length Scale

The effect of the characteristic length scale is examined by varying $l_c$ from 10$\mu$m to 500 $\mu$m as in the quiescent case. It is showing the Reynolds and Knudsen number effect hence they are directly related to the characteristic length scale. The 10 $\mu$m characteristic length corresponds to the Knudsen number of $6.23 \times 10^{-3}$, which is still in the slip flow regime. While studying the effect of the characteristic length scale, all other
design variables are at their baseline values (Table 6.1) except for the frequency, which is taken as 1 kHz.

The plot of vorticity contours is shown in Figure 6.5. Different from the quiescent case, the vorticity plot at the first stage of the synthetic jet oscillation cycle is shown in order to see the effects of the design variables more precisely. Because these vorticity plots are at the first stage, the entire maximum vorticity location is at the orifice. As shown, at the characteristic length of 500 μm a clear vortex formed and began to shed from the orifice and travels through the channel. Even though the circulating part of the vortex is around $x=2lc$, since the membrane is still pushing some of the fluid, the maximum vorticity is at the orifice. There is also another vortex in the flow domain at $x=13.6lc$. Decreasing the characteristic length yields a decrease in the formation and

![Figure 6.5 Plot of vorticity contours for different characteristic lengths](image)

Figure 6.5 Plot of vorticity contours for different characteristic lengths

shedding of the vortex structure as expected. This can also be confirmed using the vortex properties table (Table 6.3). The dimensionless metrics: the area, the circulation, and the unit circulation of the vortex decrease consistently. For $lc=400$ μm, again it is possible to
see this formation and shedding of the vortex while the vortex radius decreases a little bit. In addition, the convection velocity of the vortex (which is combination of induced velocity of the vortex and the incoming flow velocity) is decreased a little bit which can be seen visually from this plot. The second vortex and its properties are also decreased. Further increasing the characteristic length to 300 μm again yields a decrease in the vortex structure. However, this time even there is a small vortex structure at the orifice; this vortex structure is easily destroyed by the influence of cross flow and the ingestion stage of the synthetic jet. Even there is a vortex like structure at the downstream of the orifice at $x=12\text{lc}$, this is not a circulating flow structure but a vorticity value. If we further decrease the characteristic length, even these vortex-like structures disappear and the synthetic jet does not work as expected. For example for the case of $\text{lc}=10$ μm the expelled velocity (or the momentum) is so small (around two order of magnitude of the cross flow) that it is impossible to affect the cross flow.

<table>
<thead>
<tr>
<th>$\text{lc}$ (μm)</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\text{max}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>1.294</td>
<td>-0.254</td>
<td>-0.196</td>
<td>-0.671</td>
</tr>
<tr>
<td>400</td>
<td>1.102</td>
<td>-0.173</td>
<td>-0.157</td>
<td>-0.454</td>
</tr>
<tr>
<td>300</td>
<td>0.768</td>
<td>-0.093</td>
<td>-0.121</td>
<td>-0.357</td>
</tr>
<tr>
<td>200</td>
<td>0.314</td>
<td>-0.030</td>
<td>-0.095</td>
<td>-0.254</td>
</tr>
<tr>
<td>100</td>
<td>0.018</td>
<td>-0.002</td>
<td>-0.089</td>
<td>-0.124</td>
</tr>
</tbody>
</table>

The effect of the characteristic length on the orifice jet velocity profile is depicted in Figure 6.6. This figure (and all other figures) demonstrates the orifice jet velocity profile at the first stage of the synthetic jet motion similar to the vorticity plots. As in the quiescent case, there is a reverse flow at the left side of the orifice. However, this time this reverse flow is through inside the cavity while it was through the outside domain.
because the velocity profile for the quiescent case is at the third stage. As we expected the velocity increases as we increase the characteristic length.

![Graph showing velocity profiles at the orifice exit](image)

**Figure 6.6 Effect of characteristic length on velocity profiles at the orifice exit**

Figure 6.7 shows a detailed velocity and the streamline plot at the orifice. As seen from this figure, there are two circulating vortices inside the cavity. The RHS (Right Hand Side) vortex is much bigger than the LHS (Left Hand Side) one. While the membrane goes to upward, counter clockwise rotating fluid that is RHS of the orifice expelled from the orifice. This flow makes another separation bubble at the orifice. This separation bubble blocks some of the fluid. As we see in this figure, there is not an actual backflow at the orifice as seen in the velocity profile figure. Instead, the cross flow enters the orifice due to pressure drop of the sudden enlargement in the area but then goes upward together with the expelled fluid.

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The other indicator, the variation of the momentum flux is shown in Figure 6.8. As in the quiescent case, this design variable tends to increase rapidly. The maximum momentum flux value is $0.36 \text{ kg.m/s}^2$ and the minimum momentum flux value is $0.0005 \text{ kg.m/s}^2$. The difference is that Figure 5.16 is the downscale of this plot.

![Figure 6.7 Detail look up to orifice at first stage](image)

![Figure 6.8 Variation of momentum flux with respect to characteristic length](image)
6.2. Characteristic Length Scale at Constant $k_r$

In this section, we will examine the effects of the characteristic length on the vortex formation and shedding, and the momentum fluxes well as on the vortex properties for the same reduced frequency. As in the $k_r$ study in quiescent case, we will vary the characteristic length and frequency together such that the reduced frequency $k_r$ is constant. Again reduced frequency is kept constant at the value of $1.45 \times 10^3$.

When we decrease the characteristic length and increase the frequency to maintain the $k_r$ constant, there does not appear to be any visual difference between $lc=500, 400, 300 \mu m$ (Figure 6.9). Their vortex structure and the vortex locations look the same in the vorticity plot. However, if we further decrease the characteristic length, the difference in the vortex structure appears. The downstream vortex begins to disappear. For the case of $lc=100 \mu m$ this downstream vortex is diminished. There exists only one vorticity value rather than a circulating structure. For the smaller characteristic lengths even this downstream vortex structures disappear. However, the developing/shedding vortex at the orifice exists even it becomes weaker and smaller.

![Figure 6.9 Plot of vorticity contours for different characteristic lengths](image)
Investigating the vortex properties table (Table 6.4) also concludes the same results. For the \( lc=500,400,300 \) \( \mu \text{ms} \), the differences are very small. In the vortex properties table, the area and the circulation values are first increasing until \( lc=200 \) \( \mu \text{m} \) and then decreasing. This is because the vorticity becomes weaker as the characteristic length decreases and some times, it is hard to distinguish the vorticity value of the formed vortex and the vorticity value of the surroundings numerically. Nevertheless, after some characteristic length value, this surrounding vorticity value is below the cutoff number and not included in the area and circulation calculations. However, the unit vortex strength and the maximum vorticity give more valuable information. As is seen in this vortex properties table, even the difference is too small, these values, and hence the vortices are decreasing as the characteristic length decreases.

<table>
<thead>
<tr>
<th>( lc (\mu \text{m}) )</th>
<th>( S )</th>
<th>( \Gamma )</th>
<th>( \Gamma/S )</th>
<th>( \xi_{\text{max}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>1.294</td>
<td>-0.254</td>
<td>-0.196</td>
<td>-0.671</td>
</tr>
<tr>
<td>400</td>
<td>1.397</td>
<td>-0.265</td>
<td>-0.189</td>
<td>-0.597</td>
</tr>
<tr>
<td>300</td>
<td>1.533</td>
<td>-0.272</td>
<td>-0.178</td>
<td>-0.496</td>
</tr>
<tr>
<td>200</td>
<td>1.579</td>
<td>-0.253</td>
<td>-0.160</td>
<td>-0.454</td>
</tr>
<tr>
<td>100</td>
<td>1.513</td>
<td>-0.204</td>
<td>-0.135</td>
<td>-0.383</td>
</tr>
<tr>
<td>50</td>
<td>1.445</td>
<td>-0.168</td>
<td>-0.116</td>
<td>-0.341</td>
</tr>
<tr>
<td>10</td>
<td>0.359</td>
<td>-0.043</td>
<td>-0.120</td>
<td>-0.326</td>
</tr>
</tbody>
</table>

The important thing we can conclude from this vorticity contour plot as well as from the vortex properties table is that the characteristic length and the frequency are important design variables that affect the vortex structure. In addition, for the same \( k_r \), weakening by the decrease of \( lc \) is more than the strengthening by increase of the frequency. This conclusion can be understood by the variation of the momentum flux (Figure 6.10). As in the quiescent case, the increase in the momentum flux is nearly linear, which was greater than linear in the characteristic length study. If we compare the
results between quiescent and the cross flow conditions, the momentum flux relationship is linear; however, the slope of the two lines are different.

![Diagram representing the variation of momentum flux with respect to characteristic length.](image)

**Figure 6.10 Variation of momentum flux with respect to characteristic length**

### 6.3. Amplitude

In this section, the oscillation amplitude of the membrane will be examined by keeping all other design variables at the baseline values (Table 6.1). Beginning from the $A=0.2$, the value of the amplitude is changed through $A=1.6$. The plot of the vorticity contours is shown in Figure 6.11. For the value of $A=0.2$, there is a little vortex at the orifice. The other vortex in the downstream of the flow domain ($x=7.96lc$) is again a vorticity value not a circulating structure. However, in all other cases, this vorticity value is a circulating structure. Increasing the amplitude to $A=0.4$, the downstream vortex becomes more clear and the vortex at the orifice becomes thicker and stronger. The downstream vortex is at a distance of $x=10lc$. 

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 6.11 Plot of vorticity contours for different amplitudes

It is not showing in the vorticity plot but there is also a third vortex in the flow domain at $x=19lc$. Further increasing the amplitude yields similar results that are favorable effect on the vortex structure. We need to mention here that:

- The vortex structure at the orifice is changing from a circular shape ($A=0.4$) to a more stretched form.
- The counter clockwise rotating vortex at the orifice becomes more apparent as the amplitude increases.
- With the increase of the amplitude, the center of the vortex pair at the orifice move upward ($y=5.5lc$ for $A=0.2$, $y=6lc$ for $A=0.4$ $y=6.5lc$ for $A=0.8$, $y=6.8lc$ for $A=1$, and $y=7lc$ for $A=1.2$).
The distance traveled by the vortex increased from \(x=7.5lc\) for \(A=0.2\) to \(x=10lc\) for \(A=0.4\) to \(x=12lc\) for \(A=0.8\). However, after \(A=0.8\) this distance does not increase by the increase of the amplitude.

The downward vortices become clearer, stronger, and larger as the amplitude increases.

The vortex properties table (Table 6.5) is also consistent with the results that were concluded from the vorticity plot. All the vortex properties are increasing almost linearly as we increase the amplitude.

The membrane amplitude has a similar effect on the orifice jet velocity profile (Figure 6.12). Jet velocity is increasing as the amplitude increases hence the blowing of the synthetic jet also increases in this stage. Unsymmetric velocity profiles become more apparent as the amplitude increases. Figure 6.13 shows the relationship between the amplitude and the variation of the momentum flux. Similar to the quiescent case, the amplitude linearly increases the momentum that actuator expelled-ingested. However, the slope of this linear relationship is nearly twice more than that of quiescent case.

<table>
<thead>
<tr>
<th>(A/lc)</th>
<th></th>
<th>(\Gamma)</th>
<th>(\Gamma/S)</th>
<th>(\xi_{max})</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>1.951</td>
<td>-0.264</td>
<td>-0.136</td>
<td>-0.510</td>
</tr>
<tr>
<td>0.4</td>
<td>3.288</td>
<td>-0.689</td>
<td>-0.209</td>
<td>-0.868</td>
</tr>
<tr>
<td>0.8</td>
<td>5.683</td>
<td>-1.658</td>
<td>-0.292</td>
<td>-1.330</td>
</tr>
<tr>
<td>1</td>
<td>6.666</td>
<td>-2.147</td>
<td>-0.322</td>
<td>-1.788</td>
</tr>
<tr>
<td>1.2</td>
<td>7.625</td>
<td>-2.603</td>
<td>-0.341</td>
<td>-1.613</td>
</tr>
<tr>
<td>1.6</td>
<td>8.434</td>
<td>-3.419</td>
<td>-0.405</td>
<td>-4.571</td>
</tr>
</tbody>
</table>
Figure 6.12 Effect of amplitude on velocity profiles at the orifice exit

Figure 6.13 Variation of momentum flux with respect to amplitude
6.4. Frequency

Among the synthetic jet design variables, one of the key parameters is the membrane oscillation frequency. The frequency has a varying effect on the effectiveness of the synthetic jet actuator. Unlike the characteristic length or the amplitude, the frequency has a range that the synthetic jet works more efficiently. At the frequency values smaller than this range, the effect of the synthetic jet actuator is very small such that cross flow or outer domain does not sense it. For higher frequency values, the synthetic jet acts strangely and again does not affect the cross flow. In this section, the frequency study is performed by varying the frequency from 1 kHz to 200 kHz and keeping all other design variables constant at their baseline values (Table 6.1). One thing we need to mention here is that in the quiescent case, the frequency was 50 kHz to 500 kHz but now it is 1 kHz to 200 kHz. The main reason for this difference is the existence of the cross flow and hence the use of different characteristic length.

Figure 6.14 Plot of vorticity contours for different frequencies

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The vorticity contours at the first stage of the synthetic jet actuator is shown in Figure 6.14. In addition, a more detailed vorticity counter plot however, for \( f = 30, 50, 70 \) kHz including four stages of one membrane oscillation cycle is given in Figure 6.15. The vorticity plot of the smaller frequency values (1, 5, 10 kHz) are not shown in this figure because no vortex formation phenomena have been observed either in the flow domain or at the orifice. For \( f = 20 \) kHz there is a little vortex developed at the orifice and there is another vorticity value but not a circulating structure at \( x = 9l_c \). A little increase in the frequency to 25 and 30 kHz affects these vortex structures very much. The vortex at the orifice becomes more apparent, thicker, and more circular. Similar effects can also be observed for the vortex at the downstream of the flow domain.

As we come to the frequency value of 50 kHz, we can see first time the actual vortex pair that is developing at the orifice. Before this value the counter clockwise rotating vortex was always destroyed by the cross flow and hence we only see the clockwise rotating vortex at the orifice or in the flow domain. The other thing is that the vertical position of the shed vortex. In this case, the vortex is expelled through almost half of the channel and it affects not only the lower boundary but also the upper boundary of the channel. Further increasing the frequency has an unfavorable effect on the synthetic jet. As we can see from this vortex plot, at the frequency value of \( f = 80 \) kHz, there are a couple of vortex structures in the flow domain even though they are very small. There is only a small vortex for the case \( f = 100 \) kHz and finally there is no vortex or even any vortex like structures in the flow domain or at the orifice for the frequency value of \( f = 200 \) kHz. This is because, as the frequency increases, the expelled fluid does not have enough time to form a vortex. Therefore, after a small amount of fluids begins
to expel from the orifice, the suction stage of the synthetic jet already began, and this expelled fluid is drawn back into the cavity. Therefore, there is no vortex on the flow domain. Even a weaker or smaller vortex does not appear. Also for the higher frequencies, the disturbance given by the membrane changes so rapidly that this disturbance or effect of the membrane is not transferred (traveled) far enough from the membrane and then in the next stage the membrane gives an alternating disturbance (or a negative effect) and this cancels or at least decreases the previously given disturbance or effect. The synthetic jet behaves like there is no synthetic jet at all. Even the membrane pushes and pulls the fluid, there is no flow expelled or ingested from the orifice (or just very small amount of flow).

Figure 6.15 Plot of vorticity contours at four stages for $f=30$, $50$, $70$ kHz

When we look at the vortex properties table of the first vortex (Table 6.6), the vortex area, and the circulation values are increasing continuously for the first vortex
until the frequency value of 30 kHz and then it oscillates. The unit circulation continuously increases until $f=50$ kHz value and then it oscillates too. The maximum vorticity keeps increasing until $f=70$ kHz value, then there is a decrease after that point, but it is not obvious whether it is an oscillation or a decrease.

<table>
<thead>
<tr>
<th>$f$(kHz)</th>
<th>S</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{max}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0.565</td>
<td>-0.035</td>
<td>-0.062</td>
<td>-0.171</td>
</tr>
<tr>
<td>20</td>
<td>2.514</td>
<td>-0.366</td>
<td>-0.146</td>
<td>-0.497</td>
</tr>
<tr>
<td>25</td>
<td>3.288</td>
<td>-0.689</td>
<td>-0.209</td>
<td>-0.868</td>
</tr>
<tr>
<td>30</td>
<td>3.142</td>
<td>-0.821</td>
<td>-0.261</td>
<td>-1.263</td>
</tr>
<tr>
<td>50</td>
<td>0.088</td>
<td>-0.050</td>
<td>-0.565</td>
<td>-1.303</td>
</tr>
<tr>
<td>70</td>
<td>0.315</td>
<td>-0.038</td>
<td>-0.121</td>
<td>-1.823</td>
</tr>
<tr>
<td>80</td>
<td>0.013</td>
<td>-0.006</td>
<td>-0.502</td>
<td>-1.404</td>
</tr>
</tbody>
</table>

Due to the possible time delays with running different frequencies it is better to look at the second vortex on the flow domain. As seen in Table 6.7, the vortex area and the circulation peak at $f=50$ kHz. On the other hand, the unit circulation and the maximum vorticity has a maximum at $f=70$ kHz. The vortex location also peaks at this frequency value. Therefore, we may conclude that the synthetic jet works best at a frequency value between 50-70 kHz.

<table>
<thead>
<tr>
<th>$f$(kHz)</th>
<th>S</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{max}$</th>
<th>$\xi_{loc}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>1.076</td>
<td>-0.057</td>
<td>-0.053</td>
<td>-0.061</td>
<td>9.629</td>
</tr>
<tr>
<td>25</td>
<td>2.802</td>
<td>-0.231</td>
<td>-0.082</td>
<td>-0.127</td>
<td>9.862</td>
</tr>
<tr>
<td>30</td>
<td>3.473</td>
<td>-0.376</td>
<td>-0.108</td>
<td>-0.192</td>
<td>8.958</td>
</tr>
<tr>
<td>50</td>
<td>4.271</td>
<td>-0.512</td>
<td>-0.120</td>
<td>-0.271</td>
<td>11.350</td>
</tr>
<tr>
<td>70</td>
<td>3.512</td>
<td>-0.491</td>
<td>-0.140</td>
<td>-0.348</td>
<td>5.791</td>
</tr>
<tr>
<td>80</td>
<td>2.096</td>
<td>-0.277</td>
<td>-0.132</td>
<td>-0.283</td>
<td>3.903</td>
</tr>
</tbody>
</table>

This effect can be clearly seen in the orifice jet velocity profile plot (Figure 6.16a, b). The frequency plot is divided into two plots in order to be clear. The first figure (Figure 6.16a) shows the effect of the frequency on the velocity profile for the range of 1

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
to 50 kHz. In this figure, it is obviously shown that the velocity increases as the frequency increases in this range. Although there is an asymmetric profile for smaller frequencies (due to small suction in accordance with small frequency) as the frequency increases, these asymmetric behavior lessens as we go through the $f=50$ kHz value. Additionally, the reverse velocity at the left corner of the orifice exit which is apparent for $f=10$ kHz, is decreasing as we increase the frequency and finally there is no reverse flow for $f=50$ kHz. This reverse flow is due to the pressure drop when the incoming flow reaches the orifice exit. As we increase the frequency, the favorable pressure gradient provided by the synthetic jet increases and after a point it overcomes the pressure drop. For example, for $f=50$ kHz this favorable pressure gradient is more than that of the pressure drop of the incoming flow due to the orifice exit and that’s way no reverse flow observed.

\[ \text{Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.} \]
At the second figure (Figure 6.16b), the effect of the frequency on the orifice jet velocity profile is shown for the range of 50 kHz to 200 kHz. It is very clear that increasing the frequency yields a decrease in the velocity. As the frequency comes to 70 kHz, the velocity profile is like a flattened parabola and then it starts to have edge horns at both sides. These edge horns are very apparent at the frequency value of 100 kHz. Further increasing the frequency decreases the jet velocity and destructs the symmetricity of the velocity profile. Finally, for the $f=200$ kHz consistent with the vorticity plot, velocity is so small that it does not affect the outer domain. The maximum velocity is around $90 \text{ m/s}$ for $f=50$ kHz, while it is only around $5 \text{ m/s}$ for $f=200$ kHz.

Figure 6.17 shows the variation of momentum flux with respect to the frequency. There is a rapid increase in the momentum flux until $f=50$ kHz and after reaching a peak point it begins to decrease rapidly. When we compare the results for quiescent and cross
flow cases, the peak point of the momentum flux variation is around 0.6 \( \text{kg.m/s}^2 \) for cross flow case however, it was 0.35 \( \text{kg.m/s}^2 \) for quiescent case. Therefore, as seen in this plot and in the Table 6.7, the synthetic jet works best at a frequency value between 50-70 kHz.

![Graph showing variation of momentum flux with respect to frequency](image)

Figure 6.17 Variation of momentum flux with respect to frequency

6.5. Orifice Width

In this section, we will examine the effects of the orifice width. Again, all other design variables are set to their baseline values (Table 6.1). The orifice width is varied from 0.25 to 5. In order to get more accurate result, this range is divided into 10 non-overlapping intervals and the analyses are done at these points.

Figure 6.18 shows the vorticity contours for different orifice width values. For \( d_o=0.25 \), the orifice width is too small and the flow is blocked at the orifice. There is no vortex in the flow domain or at the orifice. Because of this, the \( d_o=0.25 \) case is not
shown in the vortex properties table (Table 6.8). Widening the orifice a little bit to \( d_o = 0.68 \) yields a better result. Now we can see both vortices at the flow domain and at the orifice. The value of \( d_o = 1.1 \) looks like the best choice among these points because the vortex become thicker and accomplished its development stage. The area and the circulation of the vortex increases consistent with the vorticity plot when we increase the orifice width to the \( d_o = 1.1 \). However, the unit circulation and the maximum vorticity tend to decrease as we increase the orifice width. This is because even though the flow is blocked at the orifice for small orifice width values, some of the fluid may be expelled through the orifice. Even this expelled fluid is very small amount; their vorticity value is very large which yields a stronger vortex. That is why the maximum vorticity and the unit circulation are bigger. After this point, it is observed that both of these vortices getting smaller and weaker as we increase the orifice width. This observation can also be validated through the vortex properties table (Table 6.8), which shows that when we increase the orifice width, all the vortex properties decrease after certain point. Therefore, at a value between \( d_o = 0.25 - 1.5 \), the synthetic jet works most effectively and thus it is needed to find this optimum point.

![Figure 6.18 Plot of vorticity contours for different orifice widths](image-url)
Table 6.8 Vortex properties table for orifice width study

<table>
<thead>
<tr>
<th>d_o/lc</th>
<th>S</th>
<th>( \Gamma )</th>
<th>( \Gamma/S )</th>
<th>( \xi_{max} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.6</td>
<td>1.9485</td>
<td>-0.5546</td>
<td>-0.2847</td>
<td>-1.5821</td>
</tr>
<tr>
<td>1.1</td>
<td>3.4848</td>
<td>-0.6427</td>
<td>-0.1844</td>
<td>-0.7015</td>
</tr>
<tr>
<td>1.5</td>
<td>3.3118</td>
<td>-0.4196</td>
<td>-0.1267</td>
<td>-0.3767</td>
</tr>
<tr>
<td>1.9</td>
<td>3.1320</td>
<td>-0.2971</td>
<td>-0.0949</td>
<td>-0.2124</td>
</tr>
<tr>
<td>2.4</td>
<td>2.7071</td>
<td>-0.2147</td>
<td>-0.0793</td>
<td>-0.1558</td>
</tr>
<tr>
<td>2.8</td>
<td>2.0861</td>
<td>-0.1452</td>
<td>-0.0696</td>
<td>-0.1139</td>
</tr>
<tr>
<td>3.2</td>
<td>1.6456</td>
<td>-0.0996</td>
<td>-0.0605</td>
<td>-0.0839</td>
</tr>
<tr>
<td>3.7</td>
<td>1.2562</td>
<td>-0.0670</td>
<td>-0.0533</td>
<td>-0.0695</td>
</tr>
<tr>
<td>4.1</td>
<td>0.9337</td>
<td>-0.0439</td>
<td>-0.0470</td>
<td>-0.0606</td>
</tr>
<tr>
<td>4.5</td>
<td>0.6340</td>
<td>-0.0263</td>
<td>-0.0416</td>
<td>-0.0561</td>
</tr>
<tr>
<td>5</td>
<td>0.4443</td>
<td>-0.0158</td>
<td>-0.0355</td>
<td>-0.0547</td>
</tr>
</tbody>
</table>

For the higher orifice width values than \( d_o=1.5 \), the second vortex disappeared and we only see a vorticity value not a circulating structure. This is because the orifice width is too large that the fluid inside the cavity is expelled easily and is not compressed in the cavity so their velocities are somewhat lower and not enough to form a vortex. The vortex at the orifice disappears when we further increase the orifice width to \( d_o=3.2 \). As in the case of frequency, we need to be careful about the orifice width when we design the synthetic jet. In order to use the synthetic jet effectively we need to obtain this peak value for the orifice width. Note that the vorticity plots of the cases where the orifice width larger than \( d_o=3.2 \) are not shown in this figure because of no vortex observed in the flow domain.

The effect of the velocity profile is also divided into two plots. Figure 6.19a shows the effect of orifice width on jet exit velocity profile on the first stage for the orifice with values from 0.25 to 1.9. Keep in mind that in order to compare the results, here \( x \) represents \( x=x/d_o \). First for \( d_o=0.25 \), the velocity profile has a maximum value of 28 m/s and is a reverse parabolic shape. Widening the orifice width to 0.68, yielded a 63 m/s maximum value of the velocity jump and the profile shape changed to be a blunted
reverse parabolic. For $d_o=1.1$, we can see that the velocity profile decreases and looks like that this case is worse than the $d_o=0.25$ and $d_o=0.6$ cases. This is incorrect because this plot only shows the first stage of one oscillation cycle. Four stages are included in the momentum flux variation. In addition, there occurs a reverse flow at the left hand side of the orifice exit due to the pressure drop of the incoming flow, and the favorable pressure gradient provided by the synthetic jet is not enough to recover this drop. As we increase the orifice width to 1.9, the velocities at the orifice decrease yet these reverse velocities increase. At $d_o=1.9$, the velocity profile is almost symmetric with respect to the $y=-x$ plane (Figure 6.19b). After this point the velocity profile is affected the same, as we increase the orifice width. The velocity profile for both negative and positive parts decreases continuously. At the very large value of the orifice width, the velocities are so small (maximum velocities for positive and negative part are 3.4 m/s and -5.2 m/s) such that the usual vortex structures are not observed, but flow continues to be pushed and pulled through the orifice.

This can clearly be seen in the variation of momentum flux (Figure 6.20). For the higher values of the orifice width, the momentum variation is very close to each other. After the peak point, the momentum variation decreases rapidly. However, before this peak point there is a rapid change in the momentum. The maximum momentum flux value is 0.2 kg.m/s² and minimum momentum flux value is around 0.018.
Figure 6.19 Effect of orifice width on velocity profiles at the orifice exit
6.6. Orifice Height

The orifice height is considered another design variable that might affect the synthetic jet effectiveness. For this purpose, the analyses are carried out by changing the orifice height from 0.25 to 5 as in the orifice width case. The orifice height affects the synthetic jet actuator in two ways. First, if the orifice height is long enough, the flow has a chance to become a developed flow. Secondly, if the orifice height is too long, the viscous effects through the orifice start becoming dominant.

Figure 6.21 shows the vorticity contour plot at different orifice heights. Looking at this plot carefully, we can conclude that the effect of the orifice height is not significant on the first vortex at the orifice. Because the orifice height is increasing, the distance traveled by the compressed fluid is also increasing until the expulsion from the orifice exit. We can see that in all cases, a vortex develops at the orifice; the downstream
vortex is clearly visible. However, it is possible to use an actuator effectively by designing the synthetic jet at an optimum orifice height. In the vortex plots and the vortex properties table (Table 6.8), we can see that as the orifice height increases, the vortex at the orifice seems to get smaller. Actually, this is not about the shrinking or the weakening of the vortex but also there occurs a phase shift from one stage to another.

![Figure 6.21 Plot of vorticity contours for different orifice heights](image)

This phenomenon may be verified by examining the second vortex on the flow domain. If the vortex at the orifice becomes small and weak as the orifice height increases, then this downstream vortex would also become weak or small accordingly as in previous cases. However, the second vortex is not shrinking and weakening as can be seen in these figures. A similar phenomenon occurs when we consider the distance traveled by the vortex. For this reason, the second vortex in the flow domain and its relating properties should be investigated in order to analyze the effect of the orifice height. Because the second vortices are very similar to each other in the vorticity plot,
Table 6.9 Vortex properties table of first vortex for orifice height study

<table>
<thead>
<tr>
<th>$h_{jlc}/lc$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\max}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25</td>
<td>3.3380</td>
<td>-0.6595</td>
<td>-0.1976</td>
<td>-0.7001</td>
</tr>
<tr>
<td>0.6</td>
<td>3.3360</td>
<td>-0.6708</td>
<td>-0.2011</td>
<td>-0.8096</td>
</tr>
<tr>
<td>1.1</td>
<td>3.2877</td>
<td>-0.6997</td>
<td>-0.2128</td>
<td>-0.8988</td>
</tr>
<tr>
<td>1.5</td>
<td>3.2097</td>
<td>-0.7176</td>
<td>-0.2236</td>
<td>-0.9927</td>
</tr>
<tr>
<td>1.9</td>
<td>3.0863</td>
<td>-0.7197</td>
<td>-0.2332</td>
<td>-1.0860</td>
</tr>
<tr>
<td>2.4</td>
<td>2.9648</td>
<td>-0.7066</td>
<td>-0.2383</td>
<td>-1.1711</td>
</tr>
<tr>
<td>2.8</td>
<td>2.8030</td>
<td>-0.6752</td>
<td>-0.2409</td>
<td>-1.2408</td>
</tr>
<tr>
<td>3.2</td>
<td>2.6338</td>
<td>-0.6296</td>
<td>-0.2391</td>
<td>-1.2886</td>
</tr>
<tr>
<td>3.7</td>
<td>2.2846</td>
<td>-0.5645</td>
<td>-0.2471</td>
<td>-1.3072</td>
</tr>
<tr>
<td>4.1</td>
<td>1.9567</td>
<td>-0.4943</td>
<td>-0.2526</td>
<td>-1.3025</td>
</tr>
<tr>
<td>4.5</td>
<td>1.6564</td>
<td>-0.4242</td>
<td>-0.2561</td>
<td>-1.2768</td>
</tr>
<tr>
<td>5</td>
<td>1.3596</td>
<td>-0.3553</td>
<td>-0.2614</td>
<td>-1.2333</td>
</tr>
</tbody>
</table>

Table 6.10 Vortex properties table of second vortex for orifice height study

<table>
<thead>
<tr>
<th>$h_{jlc}/lc$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\max}$</th>
<th>$\xi_{\text{loc}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25</td>
<td>2.6928</td>
<td>-0.2093</td>
<td>-0.0777</td>
<td>-0.1167</td>
<td>9.9250</td>
</tr>
<tr>
<td>0.6</td>
<td>2.7169</td>
<td>-0.2159</td>
<td>-0.0795</td>
<td>-0.1203</td>
<td>9.9250</td>
</tr>
<tr>
<td>1.1</td>
<td>2.8735</td>
<td>-0.2378</td>
<td>-0.0827</td>
<td>-0.1293</td>
<td>9.9250</td>
</tr>
<tr>
<td>1.5</td>
<td>2.9512</td>
<td>-0.2564</td>
<td>-0.0869</td>
<td>-0.1381</td>
<td>9.8081</td>
</tr>
<tr>
<td>1.9</td>
<td>3.0458</td>
<td>-0.2754</td>
<td>-0.0904</td>
<td>-0.1460</td>
<td>9.6923</td>
</tr>
<tr>
<td>2.4</td>
<td>3.1652</td>
<td>-0.2936</td>
<td>-0.0928</td>
<td>-0.1527</td>
<td>9.5775</td>
</tr>
<tr>
<td>2.8</td>
<td>3.1891</td>
<td>-0.3044</td>
<td>-0.0955</td>
<td>-0.1586</td>
<td>9.3510</td>
</tr>
<tr>
<td>3.2</td>
<td>3.2027</td>
<td>-0.3114</td>
<td>-0.0972</td>
<td>-0.1626</td>
<td>9.1287</td>
</tr>
<tr>
<td>3.7</td>
<td>3.2213</td>
<td>-0.3153</td>
<td>-0.0979</td>
<td>-0.1646</td>
<td>8.9104</td>
</tr>
<tr>
<td>4.1</td>
<td>3.1819</td>
<td>-0.3131</td>
<td>-0.0984</td>
<td>-0.1651</td>
<td>8.4857</td>
</tr>
<tr>
<td>4.5</td>
<td>3.1245</td>
<td>-0.3069</td>
<td>-0.0982</td>
<td>-0.1644</td>
<td>8.1772</td>
</tr>
<tr>
<td>5</td>
<td>3.0756</td>
<td>-0.2980</td>
<td>-0.0969</td>
<td>-0.1625</td>
<td>7.8770</td>
</tr>
</tbody>
</table>

The vortex properties table (Table 6.10) would give better results. Even these values are very close to each other and the difference is very small; all the vortex properties increase, reach a peak point, and then reduce as the orifice height increases. The vortex area and the circulation peak at 3.7; however, the unit circulation and the maximum vorticity reach a peak at 4.1. As we see in this table, the vortex location tends to decrease as we increase the orifice height.
Figure 6.22 confirms this conclusion. The jet velocity profile linearly increases by the increase of the orifice height. However, these increments are too small. The maximum velocity is 33 m/s for the smallest value of the orifice height \((h_0=0.25)\) and 62 m/s for the largest value of the orifice height \((h_0=5)\).

Variation of the momentum flux with respect to the orifice height is depicted in Figure 6.23. The momentum increases quasi-linearly until \(h_0=4.1\) and after reaching its maximum point it begins to decrease. However, as mentioned earlier, the effect of the orifice height is small. The maximum momentum flux value is 0.24 \(kg.m/s^2\) and minimum momentum flux value is around 0.19.
6.7. Cavity Height

Next, we will investigate the effect of the cavity height on the synthetic jet actuator. The range for the cavity is selected as 0.59 to 10. This time the range is divided into 15 non-overlapping intervals and the analysis are run at these points keeping all other design variables at their baseline case (Table 6.1).

In Figure 6.24 the vorticity contours are depicted. For the case of $H=0.59$ there is a little vortex developing at the orifice. However, the vortex on the downstream of the flow domain is very weak, small and is only a vorticity value rather than a circulating structure. Since the developed vortex at the orifice is weak, this weak vortex is easily destroyed by the cross flow and by the suction stage of the synthetic jet. A small increase in the cavity height to 0.67 will, affect the synthetic jet very much. The vortex at the orifice increases nearly twice and the downstream vortex becomes more apparent and
strong. The distance traveled by both vortices is increased compared to the $H=0.59$ case. For the $H=0.75$ and $H=1.0$ cases the same conclusions can be drawn. Vortices at the orifice and downstream are getting stronger and the distance traveled by these vortices is increasing (Table 6.11). However, after this point ($H=1$) increases in the cavity height adversely affect the vortex structure. The developed vortex is getting smaller and closer to the orifice at the same stage. The distance traveled by the vortices is also decreasing as the cavity height increases. Nevertheless, it is not possible to say that these shrinking vortices mean that they are weakening. Although the distance traveled by downstream vortices is decreasing, the downstream vortices do not appear to be changed in this vorticity contour plot and the vortex properties table for the second vortex (Table 6.12). This adverse effect of the cavity height continues all the way through the $H=10$ which we believe to be a value over an applicable range of cavity height. After $H=1.0$, the increment of the cavity height is around 1.8. It was 0.08 before the $H=0.1$ values. Since the differences in the vorticity plots are not large, the figures of some cases such as $H=0.83, 0.91, 1.8, 3.4, 5, 8.3$ and $9.1$ are not included.

Figure 6.24 Plot of vorticity contours for different cavity heights

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Table 6.11 Vortex properties table of first vortex for cavity height study

<table>
<thead>
<tr>
<th>$H/lc$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{max}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.59</td>
<td>1.232</td>
<td>-0.159</td>
<td>-0.129</td>
<td>-0.439</td>
</tr>
<tr>
<td>0.67</td>
<td>2.182</td>
<td>-0.398</td>
<td>-0.182</td>
<td>-0.816</td>
</tr>
<tr>
<td>0.75</td>
<td>3.059</td>
<td>-0.628</td>
<td>-0.205</td>
<td>-0.937</td>
</tr>
<tr>
<td>0.83</td>
<td>3.503</td>
<td>-0.726</td>
<td>-0.207</td>
<td>-0.893</td>
</tr>
<tr>
<td>0.91</td>
<td>3.729</td>
<td>-0.760</td>
<td>-0.204</td>
<td>-0.834</td>
</tr>
<tr>
<td>1.00</td>
<td>3.789</td>
<td>-0.767</td>
<td>-0.203</td>
<td>-0.788</td>
</tr>
<tr>
<td>1.80</td>
<td>3.490</td>
<td>-0.720</td>
<td>-0.206</td>
<td>-0.843</td>
</tr>
<tr>
<td>2.60</td>
<td>3.328</td>
<td>-0.703</td>
<td>-0.211</td>
<td>-0.743</td>
</tr>
<tr>
<td>3.40</td>
<td>3.187</td>
<td>-0.691</td>
<td>-0.217</td>
<td>-0.818</td>
</tr>
<tr>
<td>4.20</td>
<td>3.013</td>
<td>-0.670</td>
<td>-0.223</td>
<td>-0.898</td>
</tr>
<tr>
<td>5.90</td>
<td>2.610</td>
<td>-0.603</td>
<td>-0.231</td>
<td>-1.025</td>
</tr>
<tr>
<td>6.70</td>
<td>2.436</td>
<td>-0.561</td>
<td>-0.230</td>
<td>-1.059</td>
</tr>
<tr>
<td>7.50</td>
<td>2.260</td>
<td>-0.517</td>
<td>-0.229</td>
<td>-1.083</td>
</tr>
<tr>
<td>8.30</td>
<td>2.056</td>
<td>-0.469</td>
<td>-0.228</td>
<td>-1.089</td>
</tr>
<tr>
<td>9.10</td>
<td>1.952</td>
<td>-0.435</td>
<td>-0.223</td>
<td>-1.074</td>
</tr>
<tr>
<td>10.00</td>
<td>1.790</td>
<td>-0.392</td>
<td>-0.219</td>
<td>-1.077</td>
</tr>
</tbody>
</table>

Table 6.12 Vortex properties table of second vortex for cavity height study

<table>
<thead>
<tr>
<th>$H/lc$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{max}$</th>
<th>$\xi_{loc}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.67</td>
<td>1.1105</td>
<td>-0.0765</td>
<td>-0.0689</td>
<td>-0.0820</td>
<td>8.9104</td>
</tr>
<tr>
<td>0.75</td>
<td>1.9528</td>
<td>-0.1589</td>
<td>-0.0814</td>
<td>-0.1102</td>
<td>9.6923</td>
</tr>
<tr>
<td>0.83</td>
<td>2.1897</td>
<td>-0.1868</td>
<td>-0.0853</td>
<td>-0.1190</td>
<td>10.1620</td>
</tr>
<tr>
<td>0.91</td>
<td>2.2630</td>
<td>-0.1926</td>
<td>-0.0851</td>
<td>-0.1195</td>
<td>10.2820</td>
</tr>
<tr>
<td>1.00</td>
<td>2.2321</td>
<td>-0.1911</td>
<td>-0.0856</td>
<td>-0.1196</td>
<td>10.2820</td>
</tr>
<tr>
<td>1.80</td>
<td>2.1323</td>
<td>-0.1818</td>
<td>-0.0853</td>
<td>-0.1193</td>
<td>10.2820</td>
</tr>
<tr>
<td>2.60</td>
<td>2.1787</td>
<td>-0.1898</td>
<td>-0.0871</td>
<td>-0.1229</td>
<td>10.1620</td>
</tr>
<tr>
<td>3.40</td>
<td>2.2546</td>
<td>-0.1997</td>
<td>-0.0886</td>
<td>-0.1266</td>
<td>10.0430</td>
</tr>
<tr>
<td>4.20</td>
<td>2.2811</td>
<td>-0.2031</td>
<td>-0.0891</td>
<td>-0.1280</td>
<td>9.9250</td>
</tr>
<tr>
<td>5.90</td>
<td>2.2551</td>
<td>-0.2029</td>
<td>-0.0900</td>
<td>-0.1303</td>
<td>9.3510</td>
</tr>
<tr>
<td>6.70</td>
<td>2.2486</td>
<td>-0.2023</td>
<td>-0.0899</td>
<td>-0.1310</td>
<td>9.2394</td>
</tr>
<tr>
<td>7.50</td>
<td>2.1475</td>
<td>-0.1915</td>
<td>-0.0892</td>
<td>-0.1286</td>
<td>8.9104</td>
</tr>
<tr>
<td>8.30</td>
<td>2.1560</td>
<td>-0.1902</td>
<td>-0.0882</td>
<td>-0.1269</td>
<td>8.9104</td>
</tr>
<tr>
<td>9.10</td>
<td>2.0662</td>
<td>-0.1830</td>
<td>-0.0886</td>
<td>-0.1269</td>
<td>8.4857</td>
</tr>
<tr>
<td>10.00</td>
<td>2.0169</td>
<td>-0.1743</td>
<td>-0.0864</td>
<td>-0.1214</td>
<td>8.4857</td>
</tr>
</tbody>
</table>

The velocity profile at the orifice is separated into two different plots in order to compare and clarify the differences (Figure 6.25a, b). In the first plot (Figure 6.25a), $H=0.57$ to 1.8 are included. The effect of the cavity height is not well understood by this
plot. Even the reverse flow velocities at the left hand side of the orifice exit are close to each other. Positive velocities increase more than two times when we increase the cavity height from 0.59 to 0.75. However, after this point, the velocities at the right hand side of \( x=0 \) plane decrease and the velocities at the left hand side of \( x=0 \) plane increase for the \( H=0.91 \) case. For the \( H=1.8 \) case, all the velocities are decreasing however, the decrease at the right hand side is much more than that of the left hand side.

In the second figure (Figure 6.25b), the velocity profiles for cavity width from 1.8 to 10 are included. As seen in this figure, the velocities are increasing as we increase the cavity height. This seems to be a conflict with the vorticity plots and the vortex properties tables. However, this profile only shows the first stage. It is not shown here, but the decrease in the velocities in other stages is much more than the increase of the velocity profile as seen in this figure.
Figure 6.25 Effect of cavity height on velocity profiles at the orifice exit

It is better to look at the momentum variation plot in order to analyze the effect of the cavity height (Figure 6.26) because it includes four stages of one oscillation cycle. As we can see, there is a rapid change (increase) in the momentum flux from \( H=0.59 \) to \( H=0.91 \). Then reaching a peak point, the momentum tends to decrease linearly as the cavity height increases. In the range \( H=0.59-0.91 \) the change in the momentum flux is \((0.06-0.24) \text{ kg.m/s}^2\) while the decrease of the momentum flux from \( H=0.91-10 \) is \((0.24-0.217) \text{ kg.m/s}^2\).
6.8. Cavity Width

The sensitivity of the synthetic jet actuator to the cavity width is studied by varying the cavity width from 10 to 35 and setting all other design variables to their baseline values (Table 6.1).

In Figure 6.27, the vorticity contours are shown. In this figure the vorticity plots for $W=12.5$, 17.5, 22.5, 27.5 are not included since their plots does not show much difference and the plots on this figure give an idea about the cavity width effect on the vorticity. For the case of $W=10$ there is a very small vortex developing at the orifice. The second vortex, which is in the downstream of the flow domain, does not exist. However, there is a small vorticity value in the flow domain, which may represent the vortex that was previously shed from the orifice. The vortex at the orifice survives until the next stage and we can only see a vorticity value. This vorticity value also vanishes in
the next stages. Increasing the cavity width to 15 yields a better synthetic jet. The vortex at the orifice becomes thicker and stronger both visually and quantitatively (Table 6.13). The second vortex at the downstream is more noticeable, and it has a little circulating part. The distance traveled by this vortex has also increased as seen in Figure 6.27 and Table 6.14. Increasing the cavity width to 20 results in a similar effect – a stronger and thicker vortex for both at the orifice and at the downstream and more distance traveled by the downstream vortex. However, after this point, increasing the cavity width does not affect the vortex structure that much or the effect is so small that is not possible to notice in these vorticity plots. When we look at the vortex properties table, the area of the first vortex increases continuously until \( W=25 \), but then it starts to reduce while all other vortex properties are increasing as we increase the cavity width (Table 6.13). It is also true for the second vortex properties, but this time all properties including the vortex area increase continuously as we increase the cavity width (Table 6.14).

![Figure 6.27 Plot of vorticity contours for different cavity widths](image)

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Table 6.13 Vortex properties table of first vortex for cavity width study

<table>
<thead>
<tr>
<th>$W/l_c$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\text{max}}$</th>
<th>$\xi_{\text{loc}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1.8120</td>
<td>-0.2049</td>
<td>-0.1131</td>
<td>-0.3232</td>
<td></td>
</tr>
<tr>
<td>12.5</td>
<td>2.2577</td>
<td>-0.3176</td>
<td>-0.1407</td>
<td>-0.4451</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>2.6945</td>
<td>-0.4479</td>
<td>-0.1662</td>
<td>-0.5861</td>
<td></td>
</tr>
<tr>
<td>17.5</td>
<td>3.0460</td>
<td>-0.5776</td>
<td>-0.1896</td>
<td>-0.7272</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>3.2883</td>
<td>-0.6887</td>
<td>-0.2094</td>
<td>-0.8679</td>
<td></td>
</tr>
<tr>
<td>22.5</td>
<td>3.4363</td>
<td>-0.7776</td>
<td>-0.2263</td>
<td>-1.0256</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>3.4800</td>
<td>-0.8360</td>
<td>-0.2402</td>
<td>-1.1333</td>
<td></td>
</tr>
<tr>
<td>27.5</td>
<td>3.4546</td>
<td>-0.8644</td>
<td>-0.2502</td>
<td>-1.2399</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>3.3543</td>
<td>-0.8707</td>
<td>-0.2596</td>
<td>-1.3362</td>
<td></td>
</tr>
<tr>
<td>35</td>
<td>3.1694</td>
<td>-0.8473</td>
<td>-0.2673</td>
<td>-1.5144</td>
<td></td>
</tr>
</tbody>
</table>

Table 6.14 Vortex properties table of second vortex for cavity width study

<table>
<thead>
<tr>
<th>$W/l_c$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\text{max}}$</th>
<th>$\xi_{\text{loc}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>12.5</td>
<td>0.9864</td>
<td>-0.0532</td>
<td>-0.0540</td>
<td>-0.0620</td>
<td>8.4857</td>
</tr>
<tr>
<td>15</td>
<td>1.7776</td>
<td>-0.1131</td>
<td>-0.0636</td>
<td>-0.0830</td>
<td>9.0191</td>
</tr>
<tr>
<td>17.5</td>
<td>2.3684</td>
<td>-0.1733</td>
<td>-0.0732</td>
<td>-0.1058</td>
<td>9.5775</td>
</tr>
<tr>
<td>20</td>
<td>2.8017</td>
<td>-0.2308</td>
<td>-0.0824</td>
<td>-0.1270</td>
<td>9.8617</td>
</tr>
<tr>
<td>22.5</td>
<td>3.1720</td>
<td>-0.2865</td>
<td>-0.0903</td>
<td>-0.1458</td>
<td>10.1620</td>
</tr>
<tr>
<td>25</td>
<td>3.4933</td>
<td>-0.3402</td>
<td>-0.0974</td>
<td>-0.1624</td>
<td>10.2822</td>
</tr>
<tr>
<td>27.5</td>
<td>3.7101</td>
<td>-0.3821</td>
<td>-0.1030</td>
<td>-0.1747</td>
<td>10.2820</td>
</tr>
<tr>
<td>30</td>
<td>3.9416</td>
<td>-0.4215</td>
<td>-0.1069</td>
<td>-0.1848</td>
<td>10.2820</td>
</tr>
<tr>
<td>35</td>
<td>4.2036</td>
<td>-0.4733</td>
<td>-0.1126</td>
<td>-0.1976</td>
<td>10.2820</td>
</tr>
</tbody>
</table>

The effect of the cavity width on the orifice jet velocity profile is given in Figure 6.28. In this plot, the jet velocity is increasing steadily by the increase of the cavity width consistent with the vorticity plot and the vortex properties tables. While the maximum velocity for the $W=10$ case is $11 \text{ m/s}$, it jumps to $72 \text{ m/s}$ for the $W=35$ case. The reverse flow at the left hand side of the orifice exit due to the pressure drop of the incoming flow is around $-5 \text{ m/s}$ for $W=10$. As we increase the cavity width, the favorable pressure gradient provided by the synthetic jet increases. Finally, for the $W=35$ case, this pressure gradient overcomes the pressure drop of incoming flow. At this case ($W=35$), we no longer see a reverse flow at the orifice exit.

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 6.29 shows the variation of momentum flux for the cavity width. The momentum flux increases almost linearly as the cavity width increases until W=20. However, after this point even it still increases as the cavity width, the rate of the increase is reduced. In this W=10-20 range maximum to minimum momentum flux is 0.21/0.05 kg.m/s²; however, in the W=20-35 range it is 0.34/0.21 kg.m/s².
6.9. Incoming Velocity of Cross Flow

Incoming flow velocity is taken as a design variable in order to understand the response of the synthetic jet to the change in the cross flow conditions. This study should help the designer or user what to expect when the synthetic jet works under different incoming flow conditions. For this purpose, the synthetic jet actuator is simulated in different values of $U_\infty$ varying from $5 \text{ m/s}$ to $50 \text{ m/s}$.

Figure 6.30 shows the vorticity contours for different incoming flow speeds. In the first case, $U_\infty$ is fixed at $5 \text{ m/s}$. In this case, the vortex pair at the orifice clearly formed and shed from the orifice. The noticeable difference is that both of the clockwise and counter clockwise rotating vortices are formed and shed while the counter clockwise rotating vortex has not appeared in the flow domain like most of the previous cases. The other difference is the second vortex pair in the flow domain that was shed in the
previous cycle. This vortex pair traveled $5.3lc$ for the clockwise rotating vortex and $7lc$ for the counter clockwise vortex. This vortex pair makes an arc-like path and then goes downward and hits the bottom wall around $x=20lc$. This is because incoming flow speed hence the momentum is lower compared to the other cases. Then, the induced velocities of the vortices are comparable with the incoming flow. The convection velocity of two vortices, which are the combinations of induced velocity and incoming flow velocity, are not the same. However, as the time passes, the vortex is getting weaker by suction of the synthetic jet and incoming flow as well as the momentum transfer from vortex to the surroundings.

![Figure 6.30 Plot of vorticity contours for different incoming flow velocities](image)

Then the incoming flow speed is increased to $10 \text{ m/s}$. This completely changes the vortex structure in the domain. First, there is still a vortex pair at the orifice. However, the counter clockwise vortex becomes more flat. The induced velocity of the counter clockwise vortex is more than that of the clockwise rotating vortex, which yields a difference between the distances traveled by these vortices of $1lc$. This is clearly
understood by looking at the second vortices in the flow domain. Although the counter clockwise rotating vortex is just a vorticity value not a circulating structure, it represents the counter clockwise vortex that was shed in the previous cycle. It has an approximately 1.5\(l_c\) difference in the \(y\) direction and 4.5\(l_c\) in the \(x\) direction. Further increasing the \(U_\infty\) to 15 \(m/s\) or 20 \(m/s\), the clockwise rotating vortex gets close to the bottom wall and travels to downward swiping the wall. The distance traveled by the vortices in the \(y\) direction rapidly decreases as \(U_\infty\) increases. The counter clockwise rotating vortex becomes more flattened as \(U_\infty\) increases, and after one point, it is not possible to see the counter clockwise vortex (circulating flow) at the orifice or in the flow domain. As the \(U_\infty\) increases, the second vortex also becomes smaller and weaker. For example, for the \(U_\infty=30\ m/s\) case, the second vortex in the flow domain is embedded in the boundary layer, and it is hard to separate it from the boundary layer. For the \(U_\infty=40\ m/s\) and 50 \(m/s\) cases this second vortex completely disappears in the flow domain.

When we consider the vortex properties (Table 6.15), all the vortex properties are increasing except the vortex area as the incoming flow increases. The area of the vortex, however, starts to decrease after \(U_\infty=30\ m/s\). Even though the circulation still increases, after this point the increment is very small. After this particular \(U_\infty\) value, the vortex properties are less meaningful because as we increase the incoming flow, the boundary layer thickness also increases. Therefore, it is not possible to separate the vorticity value of the vortex and the vorticity value of the boundary layer.
Table 6.15 Vortex properties table for incoming flow study

<table>
<thead>
<tr>
<th>$U_\infty$</th>
<th>$S$</th>
<th>$\Gamma$</th>
<th>$\Gamma/S$</th>
<th>$\xi_{\text{max}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1.7865</td>
<td>-0.4502</td>
<td>-0.2520</td>
<td>-0.4514</td>
</tr>
<tr>
<td>10</td>
<td>1.8700</td>
<td>-0.4843</td>
<td>-0.2590</td>
<td>-0.5527</td>
</tr>
<tr>
<td>15</td>
<td>1.9726</td>
<td>-0.5298</td>
<td>-0.2686</td>
<td>-0.7364</td>
</tr>
<tr>
<td>25</td>
<td>2.1097</td>
<td>-0.6092</td>
<td>-0.2888</td>
<td>-0.9751</td>
</tr>
<tr>
<td>30</td>
<td>2.1488</td>
<td>-0.6418</td>
<td>-0.2987</td>
<td>-1.0529</td>
</tr>
<tr>
<td>40</td>
<td>2.0871</td>
<td>-0.6644</td>
<td>-0.3184</td>
<td>-1.1837</td>
</tr>
<tr>
<td>50</td>
<td>2.0014</td>
<td>-0.6660</td>
<td>-0.3328</td>
<td>-1.3007</td>
</tr>
</tbody>
</table>

In the next figure, the velocity profile at the orifice exit for different $U_\infty$ values are depicted (Figure 6.31). As shown, the left hand side of the velocity profile tends to go inward and rightward and the right hand side of the velocity profile tends to go outward and rightward as expected. In addition, as we increase the $U_\infty$, the reverse flow at the left hand side of the orifice exit increases. However, the reduction of the left hand side velocities is much more than the increase of the right hand side velocities. Therefore, this variation results in a momentum difference for different incoming flow velocities. Moreover, for the other stages, due to the orifice right side wall, the shift of the right side of the velocity profile is small compared to the shift of the left of the velocity profile. This difference also contributes to the momentum difference.

The ratio of the maximum orifice jet velocity and the incoming flow velocity varies depending on the incoming flow velocity. As seen in Figure 6.31, the maximum orifice jet velocity varies from 41 m/s (for $U_\infty=5$ m/s) to 37 m/s (for $U_\infty=50$ m/s). Therefore, the $V_{\text{jet}}/U_\infty$ ratio varies from 8.2 (for $U_\infty=5$ m/s) to 0.74 (for $U_\infty=50$ m/s) for this particular incoming flow velocity range.
The momentum flux variation with respect to the incoming flow velocity is shown in Figure 6.32. As shown, the momentum flux is decreasing linearly as the $U_\infty$ increases. Nevertheless, this decrease is very small ($0.22 \text{ kg.m/s}^2$ for $U_\infty = 5 \text{ m/s}$ and decreased to $0.19 \text{ kg.m/s}^2$ for $U_\infty = 50 \text{ m/s}$).

Figure 6.31 Effect of incoming velocity on velocity profiles at the orifice exit
6.10. Summary

Figure 6.33 is the summary of the cross flow cases. Using this plot, we can compare the design variables with each other as well. As seen, the amplitude has the largest effect, which is the same as the quiescent case. The figure does not show the values for higher amplitudes due to their large values. Again, the relationship is linear. The characteristic length also shows similar behavior with the quiescent case. The momentum flux changes rapidly by the characteristic length. The difference with the quiescent case is that the quiescent case results are downscale of cross flow results. The frequency and the orifice width also have a similar effect. For both the frequency and orifice width cases, the momentum flux, first increases then reaches a peak point and finally decreases. The cavity width has an increasing effect on the momentum in cross flow conditions; however, it was a reverse parabolic effect on the momentum for quiescent case. Moreover, the effect was small when compared to the cross flow case.
The synthetic jet push-and-pulls very small amount of momentum for very small cavity height values. This small amount of momentum increases as we increase the cavity height. However, at some point, increasing the cavity height does not affect the synthetic jet very much. As seen in Figure 6.33, the smallest effect is the effect of the orifice height, which is also an increasing-decreasing effect.

Figure 6.33 Summary of the cross flow case
7. OPTIMIZATION STUDY

Comparing figures Figure 6.8, Figure 6.13, and Figure 6.29, we observed that all these design variables have an increasing effect. Therefore, as we increase these design variables, the objective function (i.e., the momentum flux at the orifice exit) increases consistently while the incoming flow has a decreasing effect. The momentum flux decreases as we increase the incoming flow velocity. On the other hand, the frequency, the orifice width, the orifice height, and the cavity height always have an increasing-decreasing effect. They all result in a small value of momentum for their smaller values. As we increase these design variables, the momentum starts to increase, reaches a peak point and then decreases. They all provide an optimum point and one needs to find these optimum points in order to use the synthetic jet more effectively. In this section, various optimization studies are performed for orifice width, frequency, orifice height, and cavity height. There will be no optimization study for the remaining design variables such as the amplitude, the characteristic length, the cavity width, and the incoming flow velocity, because they do not show an increasing-decreasing effect. First, a one-dimensional optimization study is performed for each design variable keeping the others at their baseline values. Then an optimization study, which includes all design variables, is also performed.

Second, two types of shape optimization studies will be performed. In the first part, the design variables are the shape parameters that are the orifice width, the orifice height, and the cavity height. The other shape parameter, the cavity width, is not included in this optimization because of its increasing effect. In the first part of the
optimization studies, gradient based BFGS optimization technique embedded in the optimization code will be used in order to solve a one-dimensional unconstrained optimization problem. In the second part, the synthetic jet shape is defined by the Bezier curves. Considering the control points of the Bezier curves as a design variable, another optimization study is performed to seek the best possible synthetic jet shape yielding the most effective results. For the second problem, SQP method will be used as an optimization method.

7.1. One Dimensional Optimizations

7.1.1. Orifice Width Optimization

As seen in the cross flow and the quiescent case results, the orifice width is an important design variable that affects the synthetic jet. In Figure 6.20, which shows the variation of momentum flux with respect to the orifice width, it is seen that around a value of \( d_0 = 1 \), the synthetic jet works most effectively. To find this optimum point, a simple one-dimensional unconstrained optimization problem is constructed. All other design variables except \( d_0 \) are kept at their baseline values (Table 6.1). The optimization problem is aimed to maximize the objective function, which is the momentum flux. The only design variable is the orifice width. The optimization problem has only one side constraint for the design variable, which is \( 0.25 \leq d_0 \leq 5 \). The optimization problem is as follows:

\[
\text{Maximize } \bar{J}(d_0) \\
\text{Subject to } d_0^L \leq d_0 \leq d_0^U
\]
Optimization is performed using two different initial points in order to check if the optima reached are not local ones. The case table for the orifice width optimization is given in Table 7.1. In the first run, the initial point is selected as the baseline value, \(d^0 = 1\). At this orifice width, the resulting objective function is \(0.212 \text{ kg.m/s}^2\). After the optimization study, the optimum point is obtained as \(d^* = 0.928\) which increases the objective function slightly to \(J^* = 0.216 \text{ kg.m/s}^2\) (Figure 7.1). The increase in the objective function is not very large because the initial point is very close to the optimum point. In the second run, the initial point is selected far away from the optimum point as \(d^0 = 3\). At this orifice width value, the momentum is \(0.059 \text{ kg.m/s}^2\). After the optimization, almost the same optimum point is found, \(d^* = 0.93\) and \(J^* = 0.216 \text{ kg.m/s}^2\) (Figure 7.2). The optimization histories of the objective function and the design variable for both cases is shown in Figure 7.3 and Figure 7.4, respectively.

<table>
<thead>
<tr>
<th>Optimization Cases</th>
<th>Initial (d^0)</th>
<th>(J(d^0)) (kg.m/s²)</th>
<th>Optimal (d^*)</th>
<th>(J^<em>(d^</em>)) (kg.m/s²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.0</td>
<td>0.212</td>
<td>0.928</td>
<td>0.216</td>
</tr>
<tr>
<td>2</td>
<td>3.0</td>
<td>0.059</td>
<td>0.93</td>
<td>0.216</td>
</tr>
</tbody>
</table>

Figure 7.1 Optimization history of orifice width optimization for case 1
Figure 7.2 Optimization history of orifice width optimization for case 2

Figure 7.3 Design variable history of orifice width optimization for both cases

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The same optimum point for two different initial points convinced us that the obtained optimum is not a local optimum point, which is also verified by the momentum variation with respect to the orifice width plot (Figure 6.20).

7.1.2. Cavity Height Optimization

The other design variable that needs to be optimized is the cavity height (Figure 6.33 and Figure 6.26). The effect of the cavity height has the same characteristics with the orifice width, but the effect is even smaller when compared to the effect of the orifice width. Very small values of the orifice height affects the synthetic jet worse, however, after a reasonable value, the change in the effect is very little such that even at a very big value, \( H=10 \) (ten times larger than the baseline case) the decrease in the momentum flux is only 30%. The effect of the cavity height increases very rapidly from the smaller values and reaches the maximum point; then it reduces the objective function slowly but
steadily as we further increase the cavity height. The optimum point from Figure 6.26 again looks approximately around $H=1$. The optimization problem is as follows:

$$\text{Maximize } J(H)$$
$$\text{Subject to } H^l \leq H \leq H^u$$

This is a one-dimensional unconstrained optimization that has only one side constraint as $0.6 \leq H \leq 10$. The design variable for this problem is the cavity height and the objective function to be maximized is the momentum flux. Two different optimizations are performed in order to ensure that the found optimum is not the local optimum point. Table 7.2 is the case table for the cavity height optimization. In the first run, baseline values are used as the initial point for the design variables. At the baseline value of the cavity height $H^0=4$, the resulting objective function is $0.212 \text{ kg.m/s}^2$. At the end of the optimization run, the optimum point for the cavity height is found as $H^*=1.207$ which results in an objective function of $0.2423 \text{ kg.m/s}^2$ (Figure 7.5). This is a 12.5% increase in the objective function.

<table>
<thead>
<tr>
<th>Optimization Cases</th>
<th>Initial</th>
<th>Optimal</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$H^0$</td>
<td>$\bar{J}(H^0)$ ( kg.m/s$^2$)</td>
</tr>
<tr>
<td>1</td>
<td>4.0</td>
<td>0.212</td>
</tr>
<tr>
<td>2</td>
<td>0.7</td>
<td>0.200</td>
</tr>
</tbody>
</table>

In the second optimization run, the initial point is selected as a lower value than this optimum point, namely $H^0=0.7$. Then the optimum point is found as $H^*=1.206$ and $\bar{J}^*=0.2422 \text{ kg.m/s}^2$ (Figure 7.6). Both optimums are very close to each other and can be treated the same.
The optimization histories for the design variable and the objective function of two different optimization runs are plotted in Figure 7.7 and Figure 7.8 respectively. As seen in these figures, the optimum point is found after three iterations in the first run. However, together with the sub-iterations, analysis is called fifteen times by the optimization code. In the optimization process, gradients are requested two times. At the second run, the optimum point is found in four iterations. The analysis is called twenty times and the gradients are requested three times in the optimization process.
Figure 7.7 Design variable history of cavity height optimization for both cases

Figure 7.8 Objective function history of cavity height optimization for both cases
7.1.3. Orifice Height Optimization

As seen in Figure 6.23 and Figure 6.33, one possible optimization parameter is the orifice height. Like previous optimization studies, this is also a one-dimensional unconstrained optimization with a side constraint for the orifice height, \(0.25 \leq h_0 \leq 5\). The objective is to find the optimal orifice height dimension that gives the maximum momentum flux at the orifice. As seen in Figure 6.33, the effect of the orifice height is the smallest among the synthetic jet design variables. The lower values of the orifice height gives a minimum momentum flux of \(0.198\ \text{kg.m/s}^2\) where the maximum momentum flux in Figure 6.23 is only \(0.244\ \text{kg.m/s}^2\) at around \(h_0=4.1\). The maximum and the minimum momentum fluxes are only changed about 20%. The constructed optimization problem is in the form of:

\[
\begin{align*}
\text{Maximize} & \quad \bar{T}(h_0) \\
\text{Subject to} & \quad h_0^L \leq h_0 \leq h_0^U
\end{align*}
\]

<table>
<thead>
<tr>
<th>Optimization Case</th>
<th>Initial</th>
<th>Optimal</th>
<th>Optimal</th>
</tr>
</thead>
<tbody>
<tr>
<td>(h_0^0)</td>
<td>(\bar{T}(h_0^0)) (kg.m/s(^2))</td>
<td>(h_0^*)</td>
<td>(\bar{T}(h_0^*)) (kg.m/s(^2))</td>
</tr>
<tr>
<td>1</td>
<td>1.0</td>
<td>0.212</td>
<td>3.91</td>
</tr>
</tbody>
</table>

In the optimization run, the initial point is selected as baseline value which \(h_0^0=1\) (Table 7.3). All other design variables kept their baseline values (Table 6.1). At this point, the objective function is \(0.212\ \text{kg.m/s}^2\). The optimization code obtained the optimum point as \(h_0^*=3.91\). At this optimal point, the related objective function value is \(\bar{T}^*=0.243\ \text{kg.m/s}^2\). Because of the nature of the momentum variation with respect \(h_0\) (Figure 6.23) and being a relatively less important design variable, only one optimization has been performed for this problem. The optimization histories for the design variable
and the objective function are given in Figure 7.9 and Figure 7.10 respectively. Optimization took four iterations. In these iterations, the analysis was called fifteen times and the gradient information requested two times.

Figure 7.9 Design variable history of orifice height optimization

Figure 7.10 Objective function history of orifice height optimization
7.1.4. Oscillation Frequency Optimization

As previously mentioned, the most important design variable among all the synthetic jet design variables is the membrane oscillation frequency, because either lower or higher values of the frequency make the synthetic jet ineffective. For example, at the baseline case the momentum flux is $0.212 \text{ kg.m/s}^2$ at $f=25 \text{ kHz}$. When we decrease the frequency five times to $f=5 \text{ kHz}$, the momentum flux decreases almost twenty-three times to give $9.12 \times 10^3 \text{ kg.m/s}^2$. On the other hand, when we increase the frequency eight times to $f=200 \text{ kHz}$, the resulting momentum flux decreases almost seventeen times to give $1.25 \times 10^2 \text{ kg.m/s}^2$. At the frequency range between $f=50-70 \text{ kHz}$, the momentum flux reaches a peak point resulting a momentum more than two times larger than the baseline case. As a conclusion from these numbers, the momentum is very much sensitive to the oscillation frequency. One needs to find this optimum point in order to use the synthetic jet effectively. For this reason, a simple one-dimensional unconstrained optimization problem is constructed. The design variable this time is the membrane oscillation frequency with a side constraint of $1 \text{ kHz} < f < 200 \text{ kHz}$.

In the first optimization run, the initial point for the frequency is selected as the frequency that was used in the baseline case, which is $f=25 \text{ kHz}$. After the optimization, the optimum point is found at a value around $f^*=35 \text{ kHz}$. In the second optimization run, the initial point is selected as $f=60 \text{ kHz}$, which is a value between two maximums in the momentum variation versus the frequency plot (Figure 6.17). At this time, the optimization resulted in an optimum different from the previous optimization point. The objective function at this optimum is also larger than that of the first one. Therefore, the frequency domain has at least two local optimums. In order to examine the effect of the
frequency on the synthetic jet, a more detailed study is performed. This time, the frequency range from 1 kHz-160 kHz is divided into 60 non-overlapping, equal distance intervals (Latin Hypercube design in 1D) and the analysis are carried out at these points. However, remember that it was divided into 10 non-equally distance intervals in Section 6.4. The upper level of the frequency here is lowered from 200 kHz to 160 kHz because after 160 kHz to 200 kHz a very low momentum flux is observed. The number of points is chosen that large, in order to see the more exact distribution of momentum flux with the frequency.

The variation of the momentum flux with respect to the oscillation frequency is plotted in Figure 7.11. This is a more detailed and more points-included plot of Figure 6.17 in Section 6.4. As seen in this figure, there are a couple of local optimums in the frequency domain. The momentum flux has an oscillatory behavior with respect to the frequency where at the beginning and at the end of this range, this oscillatory behavior is damped somewhat. This plot is also interesting because to the best of the author’s knowledge, this property of the frequency on the synthetic jet has never been mentioned in the literature. There are a couple of studies that investigate the effect of the frequency on the synthetic jet; however, some of them conclude that increasing the frequency yields a better synthetic jet and while others believe that decreasing the frequency yields a better synthetic jet. There are also some studies showing that the frequency has an increasing-decreasing effect. This, we believe, is because of lack of sufficient analysis points used in these studies for the frequency domain. In Section 6.4, this increasing-decreasing effect is also found with the 10 analysis points.
The same detailed frequency study with 60 analysis points is also carried out for higher \((Kn=2.5 \times 10^{-3} \text{ for } lc=25 \ \mu m \text{ and } Kn=6.23 \times 10^{-3} \text{ for } lc=10 \ \mu m)\) and lower \((Kn=6.23 \times 10^{-3})\).
10^3 for lc=10 μm) Knudsen numbers. Surprisingly, all the frequency-vs.-momentum variation plots are very similar to each other as seen in Figure 7.12. The only difference is in their magnitudes.

In Figure 7.13 the momentum variation with respect to the frequency is shown for lc=25 μm and lc=50 μm cases. The x-axis represents the dimensionless frequency the y-axis represents the momentum flux; however, the lc=25 μm case is amplified by 2.35. As seen in this figure, both plots are very close to each other. It is also possible to show this relation with other characteristic length results.

![Figure 7.13 Momentum Variations for 25 and 50 μm](image)

Figure 7.13 Momentum Variations for 25 and 50 μm

Having a couple of local optimums to find the global optimum is an intricate problem. Some optimization techniques such as Genetic Algorithms and Simulated Annealing methods are superior by means of obtaining global optimum. However, due to their prohibitive analysis requirements and less accuracy when compared to the gradient-
based optimization methods, these algorithms are not selected in this study. On the other hand, the gradient-based optimization has possibility to be trapped at the local optimums. Therefore, to find the best solution, which is not guaranteed to be the global optimum, heavily depends on the initial point. If the initial point is somehow given close to the global optimum point, not to the local optimums, then the obtained optimum is guaranteed to be the global optimum.

Therefore, first, we need to perform some studies on the initial point to be as close as possible to the best or global optimum point. For this reason, a simple response surface (curve in one-dimension) is constructed with a given N number of points. These points are selected in the following manner: the frequency range is divided into (N-1) non-overlapping, equally distance intervals. Equal distance intervals mimic the equal probability of the points. This equal probability distribution of the points is known as the Latin Hypercube designs. Using the Least Squares method, a second order degree polynomial (Response Surface) is constructed with these given N points. Least Squares polynomial data fitting is as follows:

Suppose our polynomial is in the form of

\[ y = a_0 + a_1x + a_2x^2 \]  \hspace{1cm} (7.1)

The residual is given by:

\[ \tilde{R}^2 = \sum_{i=1}^{N} \left[ y_i - (a_0 + a_1x_i + a_2x_i^2) \right]^2 \]  \hspace{1cm} (7.2)

The partial derivatives are:
\[
\frac{\partial \tilde{R}^2}{\partial a_0} = -2 \sum_{i=1}^{N} [y_i - (a_0 + a_1 x + a_2 x^2)] = 0
\]
\[
\frac{\partial \tilde{R}^2}{\partial a_1} = -2 \sum_{i=1}^{N} [y_i - (a_0 + a_1 x + a_2 x^2)] x_i = 0
\]
\[
\frac{\partial \tilde{R}^2}{\partial a_2} = -2 \sum_{i=1}^{N} [y_i - (a_0 + a_1 x + a_2 x^2)] x_i^2 = 0
\] (7.3)

These lead to the following systems of equations

\[
a_0 \sum_{i=1}^{N} x_i + a_1 \sum_{i=1}^{N} x_i^2 + a_2 \sum_{i=1}^{N} x_i^3 = \sum_{i=1}^{N} y_i
\]
\[
a_0 \sum_{i=1}^{N} x_i^2 + a_1 \sum_{i=1}^{N} x_i^3 + a_2 \sum_{i=1}^{N} x_i^4 = \sum_{i=1}^{N} x_i^2 y_i
\] (7.4)

If we arrange them in a matrix form:

\[
\begin{bmatrix}
\sum_{i=1}^{N} x_i & \sum_{i=1}^{N} x_i^2 & \sum_{i=1}^{N} x_i^3 \\
\sum_{i=1}^{N} x_i^2 & \sum_{i=1}^{N} x_i^3 & \sum_{i=1}^{N} x_i^4 \\
\sum_{i=1}^{N} x_i^2 & \sum_{i=1}^{N} x_i^3 & \sum_{i=1}^{N} x_i^4
\end{bmatrix}
\begin{bmatrix}
a_0 \\
a_1 \\
a_2
\end{bmatrix}
= 
\begin{bmatrix}
\sum_{i=1}^{N} y_i \\
\sum_{i=1}^{N} x_i y_i \\
\sum_{i=1}^{N} x_i^2 y_i
\end{bmatrix}
\] (7.5)

Since we know the \( x_i \) and \( y_i \) points, then we can easily find the coefficients of \( a_0, a_1, a_2 \) using a simple Gauss elimination technique. Here, we used the second order degree polynomial because it has only one optimum point. This optimum point is obtained by analytically as:

\[
\frac{dy}{dx} = 0 \Rightarrow 2a_2 x + a_1 = 0 \Rightarrow x^* = -\frac{a_1}{2a_2}
\] (7.6)

This optimum point can be used as an initial point and will improve the convergence of the optimization iteration or at least more suitable than the randomly selected initial point. The relating N points and their data fitted polynomial is given in
Figure 7.14. Here the ♦ symbols are the given N points and the solid line is the data fitted polynomial. The equation of this polynomial is:

\[ y = 1.51 \times 10^{-1} + 1.69 \times 10^{-2} \, x + 1.19 \times 10^{-4} \, x^2 \]  

(7.7)

The optimum point using above equation (7.7) is:

\[ x^* = -\frac{1.69 \times 10^{-2}}{2(-1.19 \times 10^{-4})} = 70.8 \]

Figure 7.14 Original points and related data fitted curve

The second order polynomial, and hence the optimum point, can be improved by “deleting and adding” new points to the domain in the following manner. Suppose that we have been given the N number of points and their response value. When the plot of these points consist a couple of local optimums as in Figure 7.14, local optimums can be managed by checking the derivatives at each point. Since our aim is maximize the momentum, we only deal with the maximums not the minimums. After managing the
local maximums, then one point is added to the left of each maximum and another point is added to the right of the maximums in order to get more information about these local optimums. These newly added points are as follows for each $i^{th}$ maximum:

$$x_L = \frac{x_i + x_{i-1}}{2}, \quad x_R = \frac{x_i + x_{i+1}}{2}$$

Totally, $2N_{MX}^L$ points are added to the domain, where $N_{MX}^L$ is the number of local maximums. The analyses are run at these newly added points. By comparing the response values of $(N + 2N_{MX}^L)$ points, $2N_{MX}^L$ points having less response values are deleted from the frequency domain and newly improved data can be obtained. Using the same procedures, i.e. Least Square Methods, the improved data fit polynomial and hence the optimum point for this polynomial can be obtained.

This improvement procedure can be repeated as many times as desired. When repeating this procedure an infinite number of times, the optimum point of the response surface (the polynomial) will be the same as the global optimum of the frequency-vs.-momentum relation. Because every time it costs $2N_{MX}^L$ additional analysis, and this optimum point is only used as an initial point to the gradient based optimization procedure, one or two times improvement will be sufficient.

For our particular case, there are four maximums in Figure 7.14, so $2N_{MX}^L = 8$. Eight new points {25, 35, 45, 55, 65, 75, 85, and 95} are added to the domain and the analyses are run at these points again. When the response values of nineteen $(N + 2N_{MX}^L = 11 + 8 = 19)$ points are compared, the frequencies $f(\text{kHz})$={10,20,40,100,25,60,95,110} have least response value and deleted from the domain. The resulted new frequency-momentum flux variation is given together with the
old one in Figure 7.16. Here ▲ symbol represents the improved points, ♦ symbol represents the original points, long dashed line represents the improved data fit curve and solid line represents the original data fit curve. New polynomial equation is as follows:

\[ y = 1.10^{-1} + 1.86 \times 10^{-2} x + 1.36 \times 10^{-4} x^2 \]  

\[ (7.8) \]

The optimum point using above equation (7.8) is:

\[ x^* = -\frac{1.86 \times 10^{-2}}{2.(-1.36 \times 10^{-4})} = 68.4 \]

Figure 7.15 Original and the improved data points and their fitted curves

In the frequency optimization cases, these two optimal points will be used as initial points for two different cases (Table 7.4). Each of the optimizations returned the same optimum point. In the first optimization run, the initial point for the frequency is the first optimum which is \( f^0 = 70 \text{ kHz} \). After the optimization run an optimum point for the frequency is found as \( f^* = 66.9 \text{ kHz} \). This optimum point yields an optimum
objective function of $\bar{J}^* = 0.644 \text{ kg.m/s}^2$ (Figure 7.16). The optimization took four iterations. However, together with the sub iterations, the analysis was called sixteen times by the optimization code and gradient information requested three times. Moreover, we should also include the analysis that were requested to construct the response surface and improved response surface, which were eleven and eight analysis calls respectively.

<table>
<thead>
<tr>
<th>Optimization Cases</th>
<th>Initial $f^0$</th>
<th>$\bar{J}(f^0)$ (kg.m/s$^2$)</th>
<th>Optimal $f^*$</th>
<th>$\bar{J}(f^*)$ (kg.m/s$^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>70.0</td>
<td>0.583</td>
<td>66.90</td>
<td>0.644</td>
</tr>
<tr>
<td>2</td>
<td>68.4</td>
<td>0.629</td>
<td>67.03</td>
<td>0.644</td>
</tr>
</tbody>
</table>

Figure 7.16 Optimization history of frequency optimization for Case 1

In the second optimization, the improved optimal point of the improved polynomial is $f^0 = 68.4 \text{ kHz}$. This value is used as initial point at this second run. The optimal point is obtained as $f^* = 67.03 \text{ kHz}$. After the optimization this initial point yields an optimum for the objective function $\bar{J}^* = 0.644 \text{ kg.m/s}^2$ (Figure 7.17). This time
analysis is called fourteen times and gradient information is requested two times. The optimization histories of the design variable and the objective function for both optimizations are given in Figure 7.18 and Figure 7.19.

Figure 7.17 Optimization history of frequency optimization for Case 2

Figure 7.18 Design variable history for frequency optimization for both cases
7.2. Shape Optimization

In this section, results of our shape optimization studies will be presented. Two types of shape optimization will be performed. In these optimization studies, the objective is again to maximize the momentum flux at the orifice in order to use the synthetic jet most effectively. Different from the previous parametric studies and optimization studies, now we should introduce a new variable, which is used as a constraint. As mentioned in Section 6.4, at very high frequencies, the fluid does not have time to form the vortex at the orifice and thus no vortex formation or vortex-shedding phenomenon observed even the momentum flux is very large at the orifice. Considering this drawback, a new variable $St$ that is in fact the inverse of Strouhal number is introduced. The Strouhal number is a dimensionless number, which describes oscillating flow mechanisms and given as:
where, \( V \) is the velocity of the fluid, \( f \) is the oscillation frequency, and \( lc \) is the characteristic length, which is \( d_o \) in our case. The Strouhal number is used as a criterion for the vortex formation and the vortex shedding. The fluid velocity \( V \) is defined as the spatially averaged total fluid velocity during the expulsion stage. The definition is given as:

\[
V = \frac{1}{2\pi A} \int_{3\pi/2 A}^{5\pi/2} \int \bar{u}(x,t) \cdot dx \cdot d\omega,
\]

where, \( A \) is the orifice exit area, \( \omega \) \( = (3\pi/2 - 5\pi/2) \) interval corresponds to the expulsion stage, \( \bar{u} \) is the instantaneous velocity at the orifice exit. In the literature, there are some studies that suggest some formation and shedding criteria based on Strouhal number [53]. However, none of these design variables covers the microflow regimes. Due to the high frequency requirements in microflows, the aforementioned criterion does not work. Therefore in this study, we constructed an empirical criteria based on Strouhal number and used as constraint to identify how well is the synthetic jet. In our optimization studies, lower limit for this constraint is selected as 25.

In the first part, an optimization problem is constructed using the synthetic jet shape parameters, which are the orifice width, the orifice height, and the cavity height. The other shape parameter, the cavity width is not considered as a design variable due to its increasing effect on the momentum flux.

In the second part, shape of the cavity is defined using Bezier polynomials. Bezier polynomials are used to represent the complex surfaces (or curves in one-dimension). Unlike the some other representative polynomials such as interpolation
splines, the control points do not necessarily lie on the curve or surface they represent. A Bezier polynomial with \( m \) control points is an \((m-1)\)th degree Bezier polynomials. Mathematically \((m-1)\)th degree Bezier polynomials can be evaluated by the following equation:

\[
\begin{align*}
x(t) &= \sum_{i=0}^{m} \binom{m}{i} t^i (1-t)^{m-i} x_i \\
y(t) &= \sum_{i=0}^{m} \binom{m}{i} t^i (1-t)^{m-i} y_i \\
\binom{m}{i} &= \frac{m!}{i!(m-i)!}
\end{align*}
\]

where \( t \) is the parameter of the curve whose values vary uniformly between \([0, 1]\). The \((x_i, y_i)\) are the coordinates of the control points which define the curve coordinates \((x(t), y(t))\). "\( m \)" is the total number of control points representing the curve. For example, using four control points, the function of the Bezier curve would be as follows:

\[
\begin{align*}
x(t) &= x_0 (1-t)^3 + 3x_1 (1-t)^2 t + 3x_2 (1-t) t^2 + x_3 t^3 \\
y(t) &= y_0 (1-t)^3 + 3y_1 (1-t)^2 t + 3y_2 (1-t) t^2 + y_3 t^3
\end{align*}
\]

An example of this curve with four control points is given in Figure 7.20. As seen in this figure, the curve should pass through the first and the last points. In addition, at the first and the last points, the curve is tangent to the straight line passing through the first and the second points and the straight line passing through the \(m\)th and \((m-1)\)th points.
7.2.1. **Shape Optimization with Shape Parameters**

In the previous section, a one-dimensional optimizations are performed and obtained in order to use the synthetic jet most effectively. However, due to the possible nonlinear interactions of design variables with each other, using the one-dimensional optimum points does not always guarantee the most effective synthetic. On the contrary, multivariable optimization has to be performed in order to find the optimum synthetic jet shape, which yields the most effective synthetic jet. For this purpose, an optimization problem is constructed in the form of:

\[
\text{Maximize } J(d_o, h_o, H) \\
\text{Subject to } \begin{align*}
St - 25 &> 0 \\
d_o^l &\leq d_o \leq d_o^u \\
h_o^l &\leq h_o \leq h_o^u \\
H^l &\leq H \leq H^u
\end{align*}
\]

This time the objective function is a function of the orifice width, the orifice height, and the cavity height. All other design variables are kept constant at their...
baseline case values (Table 6.1) except the frequency. The optimum point obtained in Section 7.1.4 is used instead of baseline value for frequency (Table 7.4). Again, the objective function is to maximize the momentum flux at the orifice satisfying the prevailing constraints. There is only one constraint based on the Strouhal number, which identifies the synthetic jet formation and shedding phenomenon. The remaining constraints are the side constraints for each design variable which are the same as the one-dimensional case such as:

\[
\begin{align*}
0.25 & \leq d_o \leq 5 \\
0.25 & \leq h_o \leq 5 \\
0.6 & \leq H \leq 10
\end{align*}
\]

Optimization is run with a couple of different initial point configurations. In the first run the baseline values of each design variable, \(d_o^0 = 1\), \(h_o^0 = 1\), \(H^0 = 4\) selected as the initial point for the optimization. At the initial point, the objective function is 0.6437 kg.m/s². Also at the initial point \(St = 37\) so there are no violated constraints. After the optimization run, the optimum points are obtained as in Table 7.5. The objective function is increased more than 100% to give 1.393 kg.m/s². In the second run, the initial points are selected as the half of the upper bounds of each design variable. In this case, the objective function is more than that of case 1 however; the constraint value is 11.7, which is a violated constraint.

<table>
<thead>
<tr>
<th>Optimization Cases</th>
<th>Initial (d_o^0, h_o^0, H^0)</th>
<th>(\bar{J}(d_o^0, h_o^0, H^0))</th>
<th>Optimal (d_o^<em>, h_o^</em>, H^*)</th>
<th>(\bar{J}(d_o^<em>, h_o^</em>, H^*))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1, 1, 4</td>
<td>0.6437</td>
<td>1.335, 1.645, 1.595</td>
<td>1.393</td>
</tr>
<tr>
<td>2</td>
<td>2.5, 2.5, 5</td>
<td>1.103</td>
<td>1.823, 2.116, 2.117</td>
<td>1.745</td>
</tr>
</tbody>
</table>

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
After the optimization, the objective function is obtained as $1.745 \text{ kg.m/s}^2$. This is almost a 60% increase with respect to initial value and 170% increase with respect to the baseline value. Since the case 2 is the best obtained case, only the optimization results of case 2 is presented.

First, it is better to show the constraint plot in order to identify the violated optimization iterations. In Figure 7.21, the optimization history for $St$ is plotted. As seen in this figure, from the beginning to the 22nd iteration, the $St$ value is lower than the lower bound of this constraint. Therefore, until 22nd iteration, this constraint is violated. At this iteration point, the constraint value is increased by the optimization code to the value of 98, which is not shown in this plot due to its relatively large value. In the next plot, the optimization histories of the design variables are shown in Figure 7.22, Figure 7.23, and Figure 7.24 respectively.

![Figure 7.21 Optimization history for St](image)
Figure 7.22 Optimization history for design variable \( (d_0) \)

Figure 7.23 Optimization history for design variable \( (h_0) \)
Figure 7.24 Optimization history for design variable ($H$)

Figure 7.25 shows the optimization history of the objective function. As seen in Figure 7.21, first the optimization code tries to escape from the infeasible region (where there are at least one constraint is violated) even if it is necessary to decrease the objective function. As seen in this figure, the objective function is decreasing after the ninth iteration in order to escape from the infeasible region. After the 22nd iteration, the optimization this time maximizes the objective function and found the optimum point as 1.745 $kg.m/s^2$, which is 170\% increase with respect to the baseline case. The optimization report (VisualDOC Task Report) for this optimization can be seen in Appendix-C.
7.2.2. Shape Optimization with Bezier-Polynomials

In the second part, the shape optimization study will be performed using Bezier polynomials. The cavity shape is represented by Bezier curves with four control points. Of these control points, the first point is fixed and the other three points are free. The reason for fixing the first point is due to the increasing effect of the cavity width. By fixing the first point, the cavity width is fixed to its baseline value, which is 20. By using the remaining three independent control points, the shape parameters of orifice width, orifice height and the cavity height is included in the optimization. Each control point consists of two variables, which are the movement in $x$ direction and movement in $y$ direction. Therefore, there are six design variables in this optimization problem. Besides the Strouhal number constraint, there are also some geometric constraints. All these
constraints are required for the cavity to be physically meaningful. Suppose the cavity shape is as in Figure 7.26. The first constraint is about the sharpness of the orifice lip. It is defined mathematically as follows:

\[ r_{xy} = \frac{r_x}{r_y} < 5 \]

The second constraint is about the minimum gap at the orifice.

\[ d_{\text{min}} > 0.01 \]

The third constraint is about preventing the cavity from being a bizarre shape. This is done by adding an angle constraint. The angle at any point on the Bezier curve (Left Boundary (LB) and Upper Boundary (UB)) has to be less than 180° or bigger than 270°. This can be defined mathematically as follows:

\[ \bar{\theta} = \text{mod} \left[ \tan^{-1} \left( \frac{dy}{dx} \right) \cdot \frac{180}{\pi} + \theta_0, 360 \right]_{\text{LB + UB}} < 270^\circ. \]
where $\theta_0 = \begin{cases} 450 & \text{for } \Delta x > 0 \\ 270 & \text{for } \Delta x < 0 \end{cases}$

The fourth constraint is about the sharpness of the left edge. Very sharp edges are unphysical and thus undesired while obtaining the cavity shape. Mathematically, it is defined as follows:

$$ d_{yx} = \left( \frac{dy}{dx} \right)_{x=0} > 0.1 $$

The fifth constraint is about minimum distance in the $y$ direction for the cavity shape. It is undesired to be a negative $y$ value.

$$ y_{all} > 0 $$

$$ (\forall y)_{LB+UB} $$

The sixth constraint is about the minimum cavity height value. This is shown as $y_{min}$ in Figure 7.26 and should be greater than 0.2.

$$ y_{min} > 0.2 $$

$$ (\forall y)_{UB} $$

Then the optimization problem is constructed as follows:

Maximize $\bar{J}(x_1, y_1, x_2, y_2, x_3, y_3)$

$$ St - 25 > 0 $$

$$ r_{xy} < 5 $$

Subject to

$$ d_{min} < 0.01 $$

$$ \bar{\theta} < 270 $$

$$ d_{yx} > 0.1 $$

$$ y_{all} > 0 $$

$$ y_{min} > 0.2 $$

As seen from the above equation, the optimization has six design variables for four control points. These are the $x$ and $y$ movements of the control points in the

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
horizontal and vertical directions. Besides these geometric and flow type constraints, there are also side constraints for the design variables as given in Table 7.6. The initial points and the optimal results are tabulated in Table 7.7. In this optimization study, the most critical constraint is the $St$ constraint. The geometrical constraints are never violated in the optimization process. Therefore, only the $St$ constraint is plotted in Figure 7.27. The inverse-Strouhal number values are nondimensionalized by 25, which is the lower bound of this constraint. As seen in this plot, the $St$ constraint decreasing through the optimization history. However, it never violates in the optimization iterations yet some violations are observed at the sub-iterations.

Table 7.6 Side constraints for shape optimization

<table>
<thead>
<tr>
<th></th>
<th>Upper Bound</th>
<th>Lower Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>x1</td>
<td>-15.0</td>
<td>5.0</td>
</tr>
<tr>
<td>y1</td>
<td>1.0</td>
<td>4.0</td>
</tr>
<tr>
<td>x2</td>
<td>-15.0</td>
<td>5.0</td>
</tr>
<tr>
<td>y2</td>
<td>-5.0</td>
<td>4.0</td>
</tr>
<tr>
<td>x3</td>
<td>-2.0</td>
<td>-0.1</td>
</tr>
<tr>
<td>y3</td>
<td>1.0</td>
<td>8.0</td>
</tr>
</tbody>
</table>

Table 7.7 Case table for shape optimization with Bezier curves

<table>
<thead>
<tr>
<th></th>
<th>Design Variables</th>
<th>Objective Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial</td>
<td>$x1^0$ 3.5 $x2^0$ 0.5 $x3^0$ 3.0 $y1^0$ -0.5 $y2^0$ 4 $J^0 (kg.m/s^2)$</td>
<td>0.70</td>
</tr>
<tr>
<td>Optimal</td>
<td>$x1^<em>$ 2.2923 $x2^</em>$ 0.0615 $x3^<em>$ -0.2739 $y1^</em>$ -1.1469 $y2^<em>$ 6.7871 $J^</em> (kg.m/s^2)$</td>
<td>2.6853</td>
</tr>
</tbody>
</table>

In Figure 7.28, the optimization histories of all design variables are given. The fifth and the sixth design variables are increased but the remaining design variables are decreased from the initial points. Even though all the control points have an effect on the cavity shape (Bezier-Curve), the forth control point $(x3, y3)$ is in fact the most important

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
one because $x_3$ directly controls the orifice width and $y_3$ directly controls the cavity height. The orifice width is equal to two times of $x_3$. Even it is not possible to have an orifice height value as was in the previous optimization cases, the sixth design variable has a largest effect on the height of the cavity.

Figure 7.27 Optimization history of the nondimensional $St$ constraint
An example of the optimization history with and without sub iterations is given for \( x_3 \) design variable (Figure 7.29). This figure also shows the optimization history of the orifice width value. Again \( x_3 \) values are nondimensionalized with the initial point \((-0.5)\). This initial point corresponds to the \( d_0 = 1 \) value, which is the baseline case for the cross flow study. As seen in this plot, this design variable is increased to 1.14, which means the orifice width is widened to 2.3.

Finally, in Figure 7.30 the optimization history of the objective function is given. As can be seen from this figure and from the Table 7.7, the objective function is increased to 2.68 \( kg.m/s^2 \), which is almost 3 times the initial case. The resulting synthetic jet shape is given in Figure 7.31.
Figure 7.29 Optimization history of design variable ($x_3$)

Figure 7.30 Optimization history of the objective function

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
For comparison, all the obtained objective function values and the percent increased with respect to the final optimization results are tabulated in Table 7.8. Here case1 corresponds to the baseline values (Table 6.1). For this case, the objective function was obtained as $0.212 \text{ kg.m/s}^2$ in Section 6. By performing a shape optimization with Bezier curves, this momentum is increased more than 10 times. Case2 corresponds to the baseline values except the frequency. The frequency is taken as the optimum frequency value obtained in Section 7.1.4 which is 67 kHz. At this case, the objective function was found to be $0.644 \text{ kg.m/s}^2$ in Section 7.1.4. By comparing the shape optimization with Bezier curves with this case, the shape optimization results in a more than 300% increase in the objective function. Finally, the case3 corresponds to the first shape optimization study with shape parameters. Again, the shape optimization using Bezier curves resulted in a more than 50% increase in the objective function. The optimization report (VisualDOC Task Report) for this optimization can be seen in Appendix-C.
<table>
<thead>
<tr>
<th>Case</th>
<th>$J^*$ (kg.m/s²)</th>
<th>Percent Increase</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case1</td>
<td>0.212</td>
<td>1166.5</td>
</tr>
<tr>
<td>Case2</td>
<td>0.644</td>
<td>316.9</td>
</tr>
<tr>
<td>Case3</td>
<td>1.745</td>
<td>53.9</td>
</tr>
<tr>
<td>Case4</td>
<td>2.685</td>
<td></td>
</tr>
</tbody>
</table>
8. CONCLUSIONS and FUTURE WORK

8.1. Conclusions

Micro synthetic jet actuators are zero-net mass flux devices. They are small-scale devices that make them interesting for flow-control applications. One of the major advantages of the micro synthetic jets is the use of the same working fluid as the system, so they do not depend on a supply of injected fluid and relevant piping and plumbing systems. The other advantage is the minimal power requirement to drive the membrane. Furthermore, micro synthetic jets offer excellent spatial resolution and high operational frequency. The zero net mass flow is created by simply implementing the actuator. In this actuator, the actuating flow is generated at the orifice of a cavity by oscillating a membrane opposite to the orifice. Fluctuating jet flow then interacts with the outer domain and transfers linear momentum to the cross flow. A pair of vortices that forms at the orifice during the expulsion stage of the membrane oscillation cycle of the actuator then travels downward of the flow domain with the help of induced velocity and the cross flow velocity. In this study, we show a design methodology that enables us to use the micro synthetic jet actuators most effectively. Efficiency of the micro actuator highly depends on the selected design variables. This study demonstrates better or more effective use of micro actuators.

The present study consists of three major parts: parametric study of the synthetic jet in a quiescent environment, parametric study of the synthetic jet in a cross flow environment, and the optimization studies of the synthetic jet design variables. First, the
numerical simulations are designed to examine the effectiveness of a two-dimensional synthetic jet discharging into a quiescent medium by a detailed parametric study. The geometric and actuation design variables of the actuator \((W, d_m, h_m, H, A, f)\) as well as the characteristic length scale and the reduced frequency are considered. The momentum flux, jet exit velocity, vortex formation, vortex shedding, area of the vortex, the circulation of vortex are the different kinds of metrics that were used to determine the effectiveness of the synthetic jet. The following significant conclusions are drawn for the quiescent case:

- The characteristic length, \(l_c\) has greater than linear increasing effect at constant frequency. Higher \(l_c\) values result in better synthetic jet performance. The vortex formation and shedding are observed only for higher \(l_c\) values if the frequency is kept the same.

- Another characteristic length study is performed by increasing \(l_c\) while decreasing the frequency so to keep the reduced frequency constant. At this time, the vortex formation and shedding can be observed at lower values of \(l_c\). It also has a linearly changing effect on the synthetic jet, which was greater than linear for the case of characteristic length study.

- When the oscillation amplitude is considered, it is found that the momentum flux, the vortex area, and the circulation linearly increase with the amplitude. Vortex formation and vortex shedding phenomenon are highly dependent on the membrane amplitude. It is observed that higher the amplitude the more effective the synthetic jet becomes as an actuator.
• The membrane oscillation frequency, the orifice width, and the cavity width are also found to be important design variables in the effectiveness of a synthetic jet. However, one has to be more careful when considering these variables, because neither their larger nor their smaller values give the best results. On the contrary, they all provide a maximum value at some midrange. The frequency and the orifice width have big influence on the synthetic jet. Even all other design variables are at a reasonable value, if the frequency or the orifice width is not selected properly, then the synthetic jet will not work or at least its effect will be so small that the outer domain does not sense it.

• The orifice height \((h_0)\) and the cavity height \((H)\) have inverse effect. The smaller cavity and orifice heights result in a better and more effective synthetic jet.

In the second part of the study, the numerical simulations are designed to examine the effectiveness of a two-dimensional synthetic jet placed in a long microchannel. The geometric and actuation design variables of the synthetic jet are the same with the quiescent case except for the characteristic length and the frequency. Additionally, incoming flow velocity is considered as a new design variable in this case. The metrics that are used to determine the effectiveness of the synthetic jet are also the same. The following significant conclusions are drawn for the cross flow case:

• The characteristic length has greater than linear increasing effect on the momentum flux as it was in the quiescent case. Higher characteristic lengths result in better synthetic jet performance. Vortex formation and shedding are observed only for higher \(lc\) values if the frequency is kept the same. The
difference with the quiescent case is that, momentum relation of quiescent case is the downscale of that of the cross flow case. Momentum relation has a maximum to minimum value of $0.36/5.10^{-4}$ kg.m/s$^2$ for the cross flow case; however, it was $0.235/2.10^{-6}$ kg.m/s$^2$ for the quiescent case.

• The other characteristic length study, in which the reduced frequency is constant, is also performed for the cross flow case. Similar conclusions may be drawn: the vortex formation and shedding are observed at lower values of $lc$ and the momentum flux varies linearly with the characteristic length. If we compare the results between quiescent and the cross flow conditions, the momentum flux relationships are linear in both cases; however, the slope of the two lines are different.

• When the oscillation amplitude is considered, it is found that, as in the quiescent case, the amplitude linearly increases the momentum that actuator expels and ingests. However, this time the slope of this linear relationship is nearly twice more than that of the quiescent case. The maximum momentum flux for the cross flow case is around 1 kg.m/s$^2$; however, it was 0.582 kg.m/s$^2$ for the quiescent case.

• The orifice width is a key parameter in designing an actuator. For larger values of the orifice width, the fluid inside the cavity is expelled easily and is not compressed in the cavity. Therefore, their velocities are somewhat lower and not enough to form a vortex. However, for the smaller values of orifice width when the compressed fluid is leaving the orifice, the flow is blocked at the orifice hence does not generate enough momentum.
• The other key parameter is the membrane oscillation frequency, because either its lower or higher values make the synthetic jet ineffective. At the smaller values of the frequency, the membrane pushes or pulls the fluid; however, since the frequency is so small, the momentum flux in unit time is very small. The fluid inside the cavity has enough time to exit the orifice without any compression. On the other hand, for higher frequency values, the expelled fluid does not have enough time to form a vortex. Therefore, when a small amount of fluid begins to exit the orifice, the suction stage is already begun, and the expelled fluid is drawn back into the cavity. As a result, the metrics for the synthetic jet effectiveness are very small, hence, no vortex is observed in the flow domain. In addition, for the higher frequencies, the disturbance given by the membrane changes so rapidly that this disturbance or effect of the membrane is not transferred (traveled) far enough from the membrane. In the next stage, the membrane gives an alternating disturbance (or a negative effect) and this disturbance cancels or at least decreases the previously given disturbance or effect.

• The orifice height ($h_0$) and the cavity height ($H$) have an increasing-decreasing effect in the cross flow case. However, effect of the orifice height is very small when compared to the other design variables, such that, the maximum and minimum momentum rates are $0.24/0.19 \, kg.m/s^2$ for the cross flow case. For the cavity height, even very small values make the synthetic jet ineffective, and it rapidly increases the momentum. However, after a
reasonable value (H=0.75), the effect is very small and maximum to minimum value ratio is 0.24/0.17 kg/m/s².

- The cavity width had a small reverse parabolic effect on the momentum for the quiescent case; however, it has a greater than linearly increasing effect on the momentum flux for cross flow case.

- For the cross flow case and as expected, the incoming flow velocity has an unfavorable effect on the synthetic jet as expected. Although the effect of the incoming velocity is small on the momentum flux, it sufficient to destroy the vortex structure shed from the orifice.

- We may conclude from these studies that the synthetic jet discharging into a quiescent medium displays a lower limit case of what is observed for the cross flow cases. All of them respond similarly to an increment in the design variables. However, their magnitudes are different. One may expect that for the same design variables, the synthetic jet in quiescent case should expel more momentum and should work better than cross flow case.

In the last section, optimization is successfully is used to improve performance of the synthetic jet for the cross flow case. Based on the results obtained earlier, the frequency, the orifice width, the orifice height, and the cavity height are considered as the design variables for optimization. However, the characteristic length, the amplitude, and the cavity width are not included in the optimization due to their increasing effect on the synthetic jet effectiveness.

Two types of optimization studies are performed. In the first one, a one-dimensional unconstrained optimization is performed using one design variable at a time.
in order to find the optimum point, which yields the best synthetic jet. For the orifice width optimization, 2% increase is observed when compared to the baseline case. This increase may appear to be small, indicating that the baseline value of the orifice width is very close to the optimum point. With another initial point, the increase is more than 250% when compared to this initial point. On the other hand, the orifice height and the cavity height optimizations resulted in almost 15% increase in the objective function. As previously concluded, the most important design variable among all the synthetic jet design variables is the membrane oscillation frequency, because either lower or higher values of the frequency make the synthetic jet ineffective, or sometimes make it useless and wasteful.

During the frequency optimization studies, it is observed that the frequency domain has a couple of local optimums. After a detailed frequency study, it is obtained that the effect of the frequency on the synthetic jet has oscillatory behavior. At the beginning and at the end of this range, this oscillatory behavior is somehow damped. Even tough there are a couple of studies that investigate the effect of the frequency on the synthetic jets, this property of the frequency has never been shown in the literature. All of the existing studies suffer from insufficient number of analysis points, which captures only a part of this property such as increasing effect, or decreasing effect or increasing-decreasing effect.

In order to find the best solution while having many local optimums, a new methodology is introduced. First, using a response surface and least squares method an initial point for the gradient-based optimization is guessed, and then using this optimum
point as the initial point in the optimization algorithm, a new optimum point is obtained for the frequency. The optimization yields a 200% increase in the objective function.

In the second study, the shape optimization is performed to seek the best synthetic jet shape yielding the most effective results. The shape optimization is a multi variable and constrained optimization where the inverse of Strouhal number is considered as a constraint in addition to the geometrical constraints.

Two kinds of shape optimization studies are performed. In the first shape optimization study, the design variables are selected as the shape parameters, which are the orifice width, the orifice height, the cavity height. The optimization results in a 170% increase in the objective function when compared to the baseline case. In the second shape optimization study, the synthetic jet shape is defined by the Bezier curves. Considering the control points of the Bezier curves as a design variable, another optimization study is performed to find the optimum synthetic jet shape.

Satisfactory results are obtained after the optimization such that the objective function is increased more than 10 times compared to the baseline case. In addition, the optimum shape using Bezier curves results in a more than a 300% increase in the objective function as compared to the optimum frequency results and more than a 50% increase in the objective function as compared to the shape optimization using shape parameters results.

8.2. Future work

A detailed parametric study and an optimization methodology are introduced and very impressive results are obtained in this study. However, a couple of subjects should be interesting to study in the future. First, this study should be extended to an application
for both numerically and experimentally. Second, when considering the amplification or
cancellation of a vortex with another vortex, the interactions of multiple vortices may be
interesting and recommended for future study. This can be done either using multiple
synthetic jets or using pulsed synthetic jets. Third, it also should be interesting to study
further the oscillating response or performance when frequency is varied. Furthermore,
the oscillating behavior of the frequency is exciting subject. However, it is only possible
to capture such oscillations when sufficient numbers of analyses points are used.
REFERENCES


Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
[34] Reuter, J., Cliff, S., Hicks, R., and van Damm, "Practical Design Optimization of Wing/Body Configurations using Euler Equations," AIAA paper 92-2633

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.


[46] Smith, B. L., Swift, G. W., “Synthetic Jets at Large Reynolds Number and Comparison Continuous Jets ” AIAA 2001-3030


Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.


[99] Liou, W.W. and Fang, Y., Microfluid Mechanics, McGraw-Hill, New York, 2005


Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.


[121] Spalart, P. R., Almaras, S. R., “One Equation Turbulence model for aerodynamic flows,” La Recherche Aerospatiale, N.1, 1994, pp5-21

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.


APPENDIX A.

NUMERICAL METHODS FOR FLOW ANALYSIS

CFL3D is a Reynolds-averaged Navier-Stokes flow solver on structured grids. The original version of CFL3D was developed in the early 1980s in the Computational Fluids Laboratory at NASA Langley Research Center. CFL3D solves the time-dependent conservation law form of the Reynolds-averaged Navier-Stokes equations. The spatial discretization involves a semi-discrete finite-volume approach. Upwind-biasing is used for the convective and pressure terms, while central differencing is used for the shear stress and heat transfer terms. Time advancement is implicit with the ability to solve steady or unsteady flows. Multigrid and mesh sequencing are available for convergence acceleration. Numerous turbulence models are provided, including zero-equation, one-equation, and two-equation models. Multiple-block topologies are possible with the use of one-to-one blocking, patching, overlapping, and embedding. CFL3D does not contain any grid generation software. Grids must be pre-processed and supplied externally to the code. More information about the code CFL3D can be found in (Rumsey et al. [86]; Baysal et al. [87]; Bartels et al. [88]).

A.1. Governing Equations

The computational code CFL3D v6.3, which is employed for the present numerical calculations, uses the three dimensional, compressible, time dependent Navier-
Stokes equations. The set of equations can be written in terms of generalized coordinates, \((\xi, \eta, \zeta)\), as follows:

\[
\frac{\partial \hat{Q}}{\partial t} + \frac{\partial (\hat{F} - \hat{F}_v)}{\partial \xi} + \frac{\partial (\hat{G} - \hat{G}_v)}{\partial \eta} + \frac{\partial (\hat{H} - \hat{H}_v)}{\partial \zeta} = 0
\]  

(A.1)

A general, 3-dimensional transformation between the Cartesian variables \((x, y, z)\) and generalized coordinates \((\xi, \eta, \zeta)\) can be involved by using the variable \(J\) that represents the Jacobian of the coordinate transformation:

\[
J = \frac{\partial (\xi, \eta, \zeta, t)}{\partial (x, y, z, t)}
\]  

(A.2)

In equation (A.1), \(\hat{Q}\) is the vector of variables, which are density, momentum, and total energy per unit volume.

\[
\hat{Q} = \frac{Q}{J} = \frac{1}{J} \begin{bmatrix} \rho & \rho u & \rho v & \rho w & \rho e \end{bmatrix}^T
\]  

(A.3)

In equation (A.1), \(\hat{F}, \hat{G}\) and \(\hat{H}\) are the inviscid flux terms which are given as:

\[
\hat{F} = \frac{F}{J} = \frac{1}{J} \begin{bmatrix} \rho U \\ \rho U u + \xi_p \\ \rho U v + \xi_p \\ \rho U w + \xi_p \\ (\rho e + p)U - \xi_p \end{bmatrix}
\]  

(A.4)
\[
\hat{G} = \frac{G}{J} = \frac{1}{J} \begin{bmatrix}
\rho v \\
\rho v u + \eta, p \\
\rho v v + \eta, p \\
\rho v w + \eta, p \\
(\rho e + p) v - \eta, p
\end{bmatrix}
\] (A.5)

\[
\hat{H} = \frac{H}{J} = \frac{1}{J} \begin{bmatrix}
\rho w \\
\rho w u + \zeta, p \\
\rho w v + \zeta, p \\
\rho w w + \zeta, p \\
(\rho e + p) w - \zeta, p
\end{bmatrix}
\] (A.6)

where the contravariant velocities are given as:

\[
U = \xi, u + \xi, v + \xi, w + \xi, \\
V = \eta, u + \eta, v + \eta, w + \eta, \\
W = \zeta, u + \zeta, v + \zeta, w + \zeta.
\] (A.7)

Again in equation (A.1), \( \hat{F}_v, \hat{G}_v \) and \( \hat{H}_v \) are the viscous flux terms given as follows:

\[
\hat{F}_v = \frac{F_v}{J} = \frac{1}{J} \begin{bmatrix}
0 \\
\xi, \tau_{xx} + \xi, \tau_{xy} + \xi, \tau_{xz} \\
\xi, \tau_{xy} + \xi, \tau_{yy} + \xi, \tau_{yz} \\
\xi, \tau_{xz} + \xi, \tau_{yz} + \xi, \tau_{zz} \\
\xi, b_x + \xi, b_y + \xi, b_z
\end{bmatrix}
\] (A.8)

\[
\hat{G}_v = \frac{G_v}{J} = \frac{1}{J} \begin{bmatrix}
0 \\
\eta, \tau_{xx} + \eta, \tau_{xy} + \eta, \tau_{xz} \\
\eta, \tau_{xy} + \eta, \tau_{yy} + \eta, \tau_{yz} \\
\eta, \tau_{xz} + \eta, \tau_{yz} + \eta, \tau_{zz} \\
\eta, b_x + \eta, b_y + \eta, b_z
\end{bmatrix}
\] (A.9)
The shear stress and heat flux terms are defined in tensor notations as follows:

\[
\tau_{k,i,j} = \frac{M_\infty}{\text{Re}_{\lambda a}} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \lambda \frac{\partial u_k}{\partial x_i} \delta_{ij} \right] ; \quad i = 1:3, j = 1:3, k = 1:3 \quad (A.11)
\]

where;

\[
b_{ki} = u_j \tau_{k,i,j} - \dot{q}_{k,i} ; \quad i = 1:3, j = 1:3, k = 1:3 \quad (A.12)
\]

\[
\dot{q}_{k,i} = - \frac{M_\infty \mu}{\text{Re}_{\lambda a} \text{Pr} (\gamma - 1)} \left( \frac{\partial u^2}{\partial x_i} \right) \quad (A.13)
\]

Hence, the pressure is obtained by the equation of state for a perfect gas:

\[
p = (\gamma - 1) \left[ e - \frac{\rho}{2} (u^2 + v^2 + w^2) \right] \quad (A.14)
\]

The variables in the above equations are nondimensionalized with respect to the free stream density, \( \bar{\rho}_\infty \), the free stream speed of sound, \( \bar{a}_\infty \), the free stream molecular viscosity \( \bar{\mu}_\infty \) and the length \( L \).

For the thin layer approximations, the derivatives in \( \zeta \) direction, which is normal to the wall, are retained in the shear stress and heat flux terms. The pressure is nondimensionalized by the term \( \rho_\infty a_\infty^2 \). The free stream Reynolds number is defined as,

\[
\text{Re} = \frac{\rho_\infty U_\infty L}{\mu_\infty} \quad (A.15)
\]
and the Prandtl number is given by

$$Pr = \frac{\mu C_p}{k} \quad (A.16)$$

In the present study, the Prandtl number is chosen to be 0.72. The dimensionless viscosity is related to the temperature by Sutherland's law as:

$$\mu = T^\gamma \left( \frac{1 + c}{T + c} \right) \quad (A.17)$$

where $T$ is the non-dimensional temperature and $c$ is the Sutherland's constant given by $c = 110.4/T_\infty$.

A semi discrete finite volume formulation is used in the numerical algorithm of the solver for a consistent approximation to the conservation laws in integral form:

$$\frac{\partial}{\partial t} \iiint_{\Omega} Q dV + \iiint_{\partial \Omega} \vec{f} \cdot \vec{n} dS = 0 \quad (A.18)$$

Here $\vec{f}$ represents the net flux through a surface $\partial \Omega$ with unit outward normal $\vec{n}$ containing the time invariant volume $\Omega$. Integrating equation (A.18) over a control volume bounded by constant $\xi$, $\eta$ and $\zeta$ surfaces, the semi-discrete form can be obtained:

$$\left( \frac{\partial \hat{Q}}{\partial t} \right)_{i,j,k} + (\hat{F} - \hat{F}_v)_{i+1/2,j,k} - (\hat{F} - \hat{F}_v)_{i-1/2,j,k} + (\hat{G} - \hat{G}_v)_{i,j+1/2,k} - (\hat{G} - \hat{G}_v)_{i,j-1/2,k} + (\hat{H} - \hat{H}_v)_{i,j,k+1/2} - (\hat{H} - \hat{H}_v)_{i,j,k-1/2} = 0 \quad (A.19)$$
\( \Delta \xi = \xi_{i+1/2,j,k} - \xi_{i-1/2,j,k} \quad (A.20) \)

\( \Delta \eta = \eta_{i,j+1/2,k} - \eta_{i,j-1/2,k} \quad (A.21) \)

\( \Delta \zeta = \zeta_{i,j,k+1/2} - \zeta_{i,j,k-1/2} \quad (A.22) \)

In equations (A.20), (A.21) and (A.22), \( \Delta \xi, \Delta \eta \) and \( \Delta \zeta \) are taken to be equal to unity in the computational domain.

The discrete value of \( \tilde{Q}_{i,j,k} \) is taken as the average value of a unit computational cell and discrete values of \( \tilde{F}, \tilde{G} \) and \( \tilde{H} \) are regarded as face average values. Roe's upwind flux-difference splitting technique (Hirsch [89]; Hirsch [90]) was used for the convective and pressure terms and it will be discussed in the next section. The shear stress and heat transfer terms are centrally differenced.

A.2. Time Advancement

For a non-deforming mesh, Equation (A.1) can be written as:

\( \frac{1}{J} \frac{\partial \tilde{Q}}{\partial t} = R(Q) \quad (A.23) \)

where

\[
R = \left[ \frac{\partial (\tilde{F} - \tilde{F}_v)}{\partial \xi} + \frac{\partial (\tilde{G} - \tilde{G}_v)}{\partial \eta} + \frac{\partial (\tilde{H} - \tilde{H}_v)}{\partial \zeta} \right] \quad (A.24)
\]

The time term can be discretized with backward differencing:

\[
\frac{(1 + \phi)(Q^{n+1} - Q^n) - \phi(Q^n - Q^{n-1})}{J\Delta t} = R(Q^{n+1}) \quad (A.25)
\]

where the superscripts indicate time level. When \( \phi = 0 \) the method is first-order temporally accurate; when \( \phi = 1/2 \) the method is second-order accurate. This equation is...
implicit because the right-hand side is a function of the unknown flow variables at time level $n+1$.

The code is advanced in time with an implicit approximate-factorization method. The implicit derivatives are written as spatially first-order accurate, which results in block-tridiagonal inversions for each sweep. However, for the solutions that utilize FDS the block-tridiagonal inversions are usually further simplified with a diagonal algorithm (with a spectral radius scaling of the viscous terms).

Because of the method which the left-hand side is treated for computational efficiency in steady-state simulations (approximate factorization, first-order accuracy), second-order temporal accuracy is forfeited for unsteady computations. One method for recovering the desired accuracy is through the use of sub-iterations. Even though two different sub-iteration strategies have been implemented in CFL3D, “pseudo time sub­iteration (T-TS)” method is used in this study. This method is also often referred to as the “dual time stepping” method.

For the T-TS method, a pseudo time term is added to the time-accurate Navier-Stokes equations.

\[
\frac{1}{J} \frac{\partial Q}{\partial t} + \frac{(1 + \phi)(Q_{n+1} - Q^n) - \phi(Q^n - Q_n^{n+1})}{J\Delta t} = R(Q_{n+1}) \tag{A.26}
\]

This equation is then discretized and iterated in $m$, where $m$ is the sub-iteration counter.

\[
\frac{(1 + \phi')(Q_{n+1}^{n+1} - Q^n) - \phi'(Q^n - Q_n^{n+1})}{J\Delta \tau} + \frac{(1 + \phi)(Q_{n+1}^{n+1} - Q^n) - \phi(Q^n - Q_n^{n+1})}{J\Delta t} = R(Q_{n+1}^{n+1}) \tag{A.27}
\]

In equation (A.27), $\phi$ and $\phi'$ govern the order of accuracy of the physical and pseudo time terms, respectively. In practice, the pseudo time term is treated as first order.
(i.e., $\phi' = 0$), but the general form is shown here for completeness. As $m \to \infty$, the pseudo time term vanishes if the sub-iterations converge and $Q^{m+1} \to Q^{n+1}$. If $R$ is linearized with

$$R(Q^{m+1}) \equiv R(Q^{m}) + \frac{\partial R}{\partial Q} \Delta Q^{m}$$

(A.28)

and the quantity $-(1+\phi)Q^{m} / J\Delta t$ is added to both sides of equation (A.27), then equation (A.27) becomes

$$\left[\left(1+\phi' + \frac{1 + \phi}{J\Delta t}\right) I + \delta_x A + \delta_y B + \delta_z C\right] \Delta Q^{m} = \frac{\phi' \Delta Q^{m+1}}{J\Delta t} \frac{\phi \Delta Q^{n+1}}{J\Delta t} - \frac{(1 + \phi)(Q^{m} - Q^{n})}{J\Delta t} + R(Q^{m})$$

(A.29)

where

$$\Delta Q^{m} = Q^{m+1} - Q^{m}$$

$$A = \frac{\partial (\hat{F} - \hat{F}_{v})}{\partial Q}$$

$$B = \frac{\partial (\hat{G} - \hat{G}_{v})}{\partial Q}$$

$$C = \frac{\partial (\hat{H} - \hat{H}_{v})}{\partial Q}$$

Equation (A.29) is approximately factored and written in primitive variable form; it is solved as a series of sweeps in each coordinate direction.
A.3. Spatial Discretization

Discretization of inviscid fluxes

The spatial derivatives of the convective and pressure terms are written conservatively as a flux balance across a cell as, for example, the first inviscid flux in the equation (8.30) can be written as follows

\[(\delta_x \bar{F})_i = \hat{F}_{i+\frac{1}{2}} - \hat{F}_{i-\frac{1}{2}}\]  \hspace{1cm} (A.31)

where the \(i\) index denotes a cell-center location and \(i \pm \frac{1}{2}\) corresponds to a cell-interface location. The interface flux is determined from a state-variable interpolation and a locally one-dimensional flux model. Even though CFL3D code has capable of Flux limiting, Flux vector Splitting and Flux difference splitting, only the Flux Difference Splitting method is used to split these inviscid fluxes.

Flux Difference Splitting

In this technique of Roe [124], the interface flux in the \(\xi\) direction is written as

\[\bar{F}_{i+\frac{1}{2}} = \frac{1}{2} \left[ \hat{F}(q_L) + \hat{F}(q_R) - |\bar{A}_{inv}| (q_R - q_L) \right]_{i+\frac{1}{2}}\]  \hspace{1cm} (A.32)

where \(\bar{A}_{inv}\) is the evaluation of \(A_{inv}\) with Roe-averaged variables defined as below:

\[A = \frac{\partial (\bar{F} - \hat{F}_\xi)}{\partial Q}\]  \hspace{1cm} (A.33)

\[|\bar{A}_{inv}| = |A_{inv}(\tilde{q})|\]  \hspace{1cm} (A.34)

\(A_{inv}\) is the inviscid part of the matrix \(A\), that is,
\[ A_{\text{inv}} = \frac{\partial \hat{F}}{\partial Q} = T \Lambda T^{-1} = T(A^* + A^\top)T^{-1} \]  \hspace{1cm} (A.35)

\[ |A_{\text{inv}}| = T|\Lambda|T^{-1} \]  \hspace{1cm} (A.36)

Here, \( \Lambda \) is the diagonal matrix of eigenvalues of the matrix \( A_{\text{inv}} \). \( T \) is the matrix of right eigenvectors as columns and \( T^{-1} \) is the matrix of left eigenvectors as rows. They are all evaluated using Roe-averaged values such that the term given below is satisfied exactly.

\[ \hat{F}(Q_R) - \hat{F}(Q_L) = |A_{\text{inv}}|(Q_R - Q_L) \]  \hspace{1cm} (A.37)

Here, the term \( |A_{\text{inv}}|(Q_R - Q_L) \) can be written as below:

\[
|A_{\text{inv}}|(Q_R - Q_L) \equiv |\tilde{A}_{\text{inv}}| \Delta Q = \begin{bmatrix}
\alpha_4 \\
\tilde{u} \alpha_4 + \tilde{\xi}_5 \alpha_5 + \alpha_6 \\
\tilde{u} \alpha_4 + \tilde{\xi}_5 \alpha_5 + \alpha_7 \\
\tilde{u} \alpha_4 + \tilde{\xi}_5 \alpha_5 + \alpha_8 \\
\tilde{H} \alpha_4 + (\tilde{K} - \tilde{\xi}_7) \alpha_5 + \tilde{u} \alpha_6 + \tilde{v} \alpha_7 + \tilde{w} \alpha_8 - \frac{\tilde{a}^2 \alpha_1}{\gamma - 1}
\end{bmatrix}
\]  \hspace{1cm} (A.38)

where,
\( \alpha_1 = \left( \frac{\nabla \xi}{J} \right) \tilde{U} \left( \Delta \rho - \frac{\Delta p}{\tilde{a}^2} \right) \)
\( \alpha_2 = \frac{1}{2\tilde{a}^2} \left( \frac{\nabla \xi}{J} \right) \tilde{U} + \tilde{a} \left( \Delta \rho + \tilde{p} \tilde{a} \Delta \tilde{U} \right) \)
\( \alpha_3 = \frac{1}{2\tilde{a}^2} \left( \frac{\nabla \xi}{J} \right) \tilde{U} - \tilde{a} \left( \Delta \rho - \tilde{p} \tilde{a} \Delta \tilde{U} \right) \)
\( \alpha_4 = \alpha_1 + \alpha_2 + \alpha_3 \)
\( \alpha_5 = \tilde{a} (\alpha_2 - \alpha_3) \)
\( \alpha_6 = \left( \frac{\nabla \xi}{J} \right) \tilde{U} \left( \tilde{p} \Delta u - \xi, \tilde{p} \Delta \tilde{U} \right) \)
\( \alpha_7 = \left( \frac{\nabla \xi}{J} \right) \tilde{U} \left( \tilde{p} \Delta u - \xi, \tilde{p} \Delta \tilde{U} \right) \)
\( \alpha_8 = \left( \frac{\nabla \xi}{J} \right) \tilde{U} \left( \tilde{p} \Delta u - \xi, \tilde{p} \Delta \tilde{U} \right) \)

The tilde (~) symbol indicates the following Roe-averaged variables.

\[ \tilde{\rho} = \sqrt{\rho_L \rho_R} \]
\[ \tilde{u} = \frac{u_L + u_R \sqrt{\rho_R / \rho_L}}{1 + \sqrt{\rho_R / \rho_L}} \]
\[ \tilde{v} = \frac{v_L + v_R \sqrt{\rho_R / \rho_L}}{1 + \sqrt{\rho_R / \rho_L}} \]
\[ \tilde{w} = \frac{w_L + w_R \sqrt{\rho_R / \rho_L}}{1 + \sqrt{\rho_R / \rho_L}} \]
\[ \tilde{H} = \frac{H_L + H_R \sqrt{\rho_R / \rho_L}}{1 + \sqrt{\rho_R / \rho_L}} \]
\[ \tilde{a}^2 = (\gamma - 1) \tilde{H} - \tilde{u}^2 + \tilde{v}^2 + \tilde{w}^2 \]

and,
\[ \tilde{U} = \frac{1}{|\nabla \xi|} (\xi_x \tilde{u} + \xi_y \tilde{v} + \xi_z \tilde{w} + \xi_t) \]
Discretization of Viscous Fluxes

The viscous terms, which represent shear stress and heat transfer effects, are discretized with second-order central differences. The second derivatives are treated as differences across cell interfaces of first-derivative terms. Hence, in the direction for example, the viscous terms are discretized as

\[(\delta_\xi \hat{F}_v)_i = (\hat{F}_v)_{i+\frac{1}{2}} - (\hat{F}_v)_{i-\frac{1}{2}}\]  \hspace{1cm} (A.42)

The term \(\hat{F}_v\) is given in Equation (A.8). Using the thin-layer approximation:

\[
\hat{F}_v = \frac{M - \mu}{\text{Re} \xi} \frac{1}{J} \begin{bmatrix}
0 \\
\phi_1 \frac{\partial u}{\partial \xi} - \xi_1 \phi_2 \\
\phi_1 \frac{\partial v}{\partial \xi} - \xi_1 \phi_2 \\
\phi_1 \frac{\partial w}{\partial \xi} - \xi_1 \phi_2 \\
\phi_1 \left[ \frac{1}{2} \left( \frac{V}{J} \right)^2 \right] + \frac{1}{\text{Pr}(\gamma - 1)} \left[ \frac{1}{2} \right] + (U - \xi_1) \phi_2
\end{bmatrix} \]  \hspace{1cm} (A.43)

where

\[
\phi_1 = \xi_x^2 + \xi_y^2 + \xi_z^2 \\
\phi_2 = \left( \frac{\phi_x \frac{\partial u}{\partial \xi} + \frac{\phi_y}{\partial \xi} + \frac{\phi_z}{\partial \xi}}{3} \right)
\]

A.4. Moving Deforming Mesh

CFL3D has the capability to perform computations for prescribed surface motion in two ways. One of them is the (or user specified) rigid grid motion. In this mode, the entire grid or set of grids translates or rotates in a manner prescribed by user input. The other one is the prescribed surface motion with deforming mesh. In this mode, the surface(s) prescribed by the user translate or rotate and the mesh deforms accordingly. In
the present study, the prescribed surface motion with deforming mesh is used to accurately model the synthetic jets membrane motion.

CFL3D can perform several types of user specified surface motion by deforming the mesh, i.e. surface rotation and/or translation of all or partial segments of the solid surfaces as well as modal motion of surfaces. Aero-elastic, user defined deforming mesh surface and user defined rigid grid motion can be performed in any combination. There are two methods of deforming the mesh. The first one is the exponential decay combined with Trans-Finite Interpolation (TFI) of interior mesh points. The second one is the finite macro-element deformation combined with TFI.

In the first mesh movement option, (exponential decay method) deformation is performed in two steps. The first step is exponential decay of control points away from the moving surface. The rate of the exponential decay is controlled by user input. The second step is a TFI of mesh point's interior to the control points. Advantage of the exponential decay method is that it is computationally efficient.

In the second mesh movement option, (finite macro-element method) deformation is also performed in two steps. The first step is a finite element solution of macro-element points. The resulting solution transmits surface motion to the element node points. The element stiffness varies with distance from the surface. User specified input controls the rate at which the element stiffness decays away from surfaces. The second step is a TFI of mesh point's interior to the element node (or control) points. Advantage of the finite macro-element method is that it maintains mesh quality, but is significantly more computationally time consuming. See references (Rumsey et al. [86]; Baysal et al. [87]; Bartels et al. [88]) for more details.
APPENDIX B.

MESH QUALITY IMPROVEMENT

As mentioned in Section 3.1, in the shape optimization studies using Bezier-polynomials, the grid required by the analysis code sometimes becomes more complicated and it is not possible to generate a good-quality grid using a simple FORTRAN algorithm. One way to overcome this drawback is to use the powerful, commercial grid generation softwares such as GRIDGEN. However, they are user-dependent which is at each optimization iteration grid has to be supplied externally. Externally providing grid to the optimization algorithm is too much time consuming, complicated and non-scientific. Yet an automated grid generation technique is required in order to perform the optimization efficiently. For this reason a FORTRAN grid generation algorithm is developed. This algorithm provides an initial grid, which is very poor usually. Then using the Mesh Quality Improvement Toolkit (MESQUITE) [111], the quality of the mesh is improved and supplied to the analysis code to perform analysis and optimization.

Mesquite is a stand-alone, portable mesh improvement toolkit that can be easily linked via functional interfaces to many different mesh generation and application codes. It uses the state of the art optimization algorithms to improve homogeneous simplicial meshes. More detail about the mesquite can be found in [111], [112], [113].

Mesquite software design is based on a mathematical framework that improves the mesh quality by solving an optimization problem to guide the movement of mesh vertices. The user inputs a mesh or sub-mesh consisting of vertices, elements and the
relationship between them. The quality of each vertex or element in the mesh is described by a local quality metric that is a function of a subset of the mesh vertices. The global quality of the mesh is performed by taking the global norm of the local mesh qualities. The global quality is thus a function of the positions of all the mesh vertices. If this function can be used in a well-posed minimization problem, mesh vertices are moved by mesquite toward the vertex positions of the optimal mesh, thus improving the quality according to the criterion defined by the local quality metric. By changing the local quality metric, one can archive a variety of mesh quality improvement goals such as shape improvement and size adaptation. Suppose a mesh that contains \( n \) vertices and \( k \) elements. Then the quality of each element or vertex is given as a general function \( q_i(x) \) of the coordinates of the vertex locations. A mesh quality objective function \( \tilde{J} = f(q(x)) \) for \( i=1, n_s \) is formed to give overall measure of the mesh quality where \( n_s \) is the number of vertices in a mesh sub-domain needing improvement. Mesquite is design to solve the minimization problem \( \min \tilde{J} \) for a broad collection of the quality metrics \( q_i(x) \) and the objective function \( \tilde{J} \) using different types of optimization techniques.

Four main topics of the mesh quality improvement are the quality metrics, the objective functions, the quality improvers, and the termination criteria.

**Quality metric**

In mesquite, the quality metric provides a measure of the quality of individual mesh entities. The quality metrics can evaluate either the element quality or vertex quality. There are currently a couple of quality metrics available within mesquite such as Mean Ratio, Area Smoothness, Aspect Ratio, Condition Number, Corner Jacobian, and Edge Length. These metrics are applicable for any meshes except the Aspect Ratio.
metrics, which works only for triangle and tetrahedral elements. For more details refer to [111], [112], [113], and [124]. There are two composite metrics that allow to use of two metric values together or to use a single metric value with a power. The latter also allows negative powers to obtain inverse of any quality metric.

Based on the recommendations in the user guide [113], the inverse mean ratio metric is used in this study, which has been extensively optimized, and analytical gradients and Hessians are available for this metric. For more information about the mean ratio metric, its construction, convexity, and usage in the mesquite can be found in [113], [124], and [126].

**Objective functions**

While the quality metric provides quality information of individual mesh entities, the objective function provides combination of these quality information values into a single number for the optimization problem. For example, $L_\infty$ norm will tend to improve the worst-case local quality while a $L^2_2$ norm will improve the RMS quality of the global mesh. The latter objective function is the square of standard $L^2_2$ norm. Given a quality metrics $q_i(x)$, the mesh quality objective function would be:

$$J = f(q(x)) = \sum_{i=1,n}(q_i(x))^2$$

The gradient and the Hessian information of this function required for the optimization process. These are computed either analytically (if using the mean ratio quality metric for example) or numerically. Numerical computation is computationally expensive however, only needs the quality metric values.
Quality improvers

Quality improvers are the optimization algorithms that move the vertex of each element to improve the meshes. Currently available optimization algorithms in the mesquite are the conjugate gradient method, the feasible Newton algorithm, and the active set algorithm. Conjugate gradient algorithm is appropriate for optimizing any combination of $C^1$ objective functions and quality metrics. It has a linear convergence property for most of the problems the algorithm requires the objective function value and gradient information. The feasible Newton method minimizes a quadratic approximation of a nonlinear objective function. It is known to converge super-linearly near local minimum. It requires the objective function value, gradient and Hessian information. When applicable it is recommended as a worthwhile choice [125].

The active set algorithm has been developed for non-continuously differentiable objective functions. Currently this algorithm works on a patch of triangular or tetrahedral elements that contain single free vortex. For more details, see [112] and [113].

Termination criteria

Wide variety of termination criteria can be selected singly or in combination to tell the solver when to stop. There are a couple of termination criteria in mesquite such as the maximum number of iterations, maximum allotted time or a sufficiently small objective function gradient

An Example

The synthetic jet cavity shape is constructed using Bezier polynomials. Using four Bezier control points the computational domain is given in Figure B.1.
This grid is created using a simple FORTRAN code and given to the mesquite as an initial grid. Selecting the inverse mean ratio as a quality metric and $L_2^2$ norm as an objective function, mesquite program gave the following output. As seen in this output the average inverse mean ratio is 20, RMS value is 753 and this initial grid has a maximum inverse mean ratio of 46167. After the optimization to reduce the inverse
mean ratio, the average is reduced to 7.41, RMS reduced to 14.55 and the maximum valued inverse mean ratio is reduced to 783. The smoothed mesh for zoomed region is given in Figure B.2.

Working with mesh file: channel.vtk

************** QualityAssessor Summary **************

There were no inverted elements detected.
No entities had undefined values for any computed metric.

<table>
<thead>
<tr>
<th>metric</th>
<th>minimum</th>
<th>average</th>
<th>rms</th>
<th>maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inverse Mean Ratio</td>
<td>1.01535</td>
<td>20.0263</td>
<td>753.949</td>
<td>49167.7</td>
</tr>
</tbody>
</table>

************** QualityAssessor Summary **************

There were no inverted elements detected.
No entities had undefined values for any computed metric.

<table>
<thead>
<tr>
<th>metric</th>
<th>minimum</th>
<th>average</th>
<th>rms</th>
<th>maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inverse Mean Ratio</td>
<td>1.00197</td>
<td>7.41381</td>
<td>14.5556</td>
<td>783.254</td>
</tr>
</tbody>
</table>
This smoothed mesh is again given as initial mesh to the mesquite and the optimization re-run. The output of the run is given as below. Even the minimum inverse mean ratio is increased a little bit from 1 to 1.41. Average, RMS, and the maximum inverse mean ratio values are decreased very much and outputs a grid as in Figure B.3. After this point, providing the smoothed mesh to the mesquite to repeat the optimization does not yield any difference in the grid quality.
Working with mesh file: smoothed_mesh.vtk

************** QualityAssessor Summary **************

There were no inverted elements detected.

No entities had undefined values for any computed metric.

<table>
<thead>
<tr>
<th>metric</th>
<th>minimum</th>
<th>average</th>
<th>rms</th>
<th>maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inverse Mean Ratio</td>
<td>1.00197</td>
<td>7.41727</td>
<td>14.7384</td>
<td>797.083</td>
</tr>
</tbody>
</table>

************** QualityAssessor Summary **************

There were no inverted elements detected.

No entities had undefined values for any computed metric.

<table>
<thead>
<tr>
<th>metric</th>
<th>minimum</th>
<th>average</th>
<th>rms</th>
<th>maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inverse Mean Ratio</td>
<td>1.41051</td>
<td>3.4484</td>
<td>3.58836</td>
<td>10.2201</td>
</tr>
</tbody>
</table>

Figure B.3 Final grid zoomed inside the left corner
APPENDIX C.
VISUALDOC TASK REPORTS

VisualDOC task report of the cavity shape optimization with shape parameters in Section 7.2.1 for case2 (Table 7.5) is given below. In this report, the sub iteration results and the final results for first, second, third, fifth and sixth are not included.

VisualDOC Task Report

==============================================================================
* Report Version : 3.00 (0)
* Database Version : 3.00 (0)
* Database File Name : /.../shp3
* Task Number : 12
  * Task Name : f67-cnstrnt-2.5-2.5-5
  * Run Status : Completed
==============================================================================

DESIGN TASK CONTROL INFORMATION

==============================================================================
f67-cnstrnt-2.5-2.5-5
Nonlinear, Constrained Optimization

Task Database ID : 12
Optimization Method : SQP
Print Control : None
  Objective : Maximize
Constraint Tolerance : -0.03
Violated Constraint Tolerance : 0.003
Gradients Calculated By : First Forward Difference
Relative Finite Difference Step : 0.001
Minimum Finite Difference Step : 0.0001
-----------------------------------------------------------------------------
Optimizer Parameters
Relative Hard Convergence Criteria : 0.001
Absolute Objective Convergence Tol. : Default
Consecutive Iter. for Convergence : 2
Iter. b/w Design Variable Scaling : Default
Max. Number of Iter. by the Optimizer : 100
-----------------------------------------------------------------------------
SLP/SQP Sub-Problem
  Maximum Iterations : 50
  Consecutive Iter. for Convergence : 2
-----------------------------------------------------------------------------

Independent Design Variables : 3
Linked Design Variables : 1
Synthetic Design Variables : 0
Constant Design Variables : 0
Independent Responses : 2

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Linked Responses : 0
Synthetic Responses : 0
Constraints : 1
Objectives : 1
Link Specifications : 1
Synthetic Function Specifications : 0
Number of Discrete Set Specifications : 0

Analysis Program : /.../shp3.pl
Design Variable Transfer : dvar.vef
Responses Transfer : resp.vef

DESIGN VARIABLE INFORMATION

<table>
<thead>
<tr>
<th>Transfer Sequence</th>
<th>Type</th>
<th>Name</th>
<th>DB ID</th>
<th>Auxil DB ID</th>
<th>Lower Bound</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 IDVar</td>
<td>d</td>
<td>1</td>
<td>0.2500000</td>
<td>5.000000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2 IDVar</td>
<td>h</td>
<td>2</td>
<td>0.2500000</td>
<td>5.000000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3 IDVar</td>
<td>H</td>
<td>3</td>
<td>0.7500000</td>
<td>10.000000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4 LDVar</td>
<td>lref</td>
<td>4</td>
<td>1 -1.000000e+30</td>
<td>1.000000e+30</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

LINKED DESIGN VARIABLES

<table>
<thead>
<tr>
<th>DVar Name</th>
<th>Link Name</th>
<th>Arguments</th>
</tr>
</thead>
<tbody>
<tr>
<td>lref</td>
<td>Lnk_l</td>
<td>d</td>
</tr>
</tbody>
</table>

DESIGN RESPONSE INFORMATION

<table>
<thead>
<tr>
<th>Transfer Sequence</th>
<th>Type</th>
<th>Name</th>
<th>DB ID</th>
<th>Auxil DB ID</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 IResp</td>
<td>mflx</td>
<td>1</td>
<td>none</td>
<td></td>
</tr>
<tr>
<td>2 IResp</td>
<td>st</td>
<td>2</td>
<td>none</td>
<td></td>
</tr>
</tbody>
</table>

DESIGN CONSTRAINT INFORMATION

<table>
<thead>
<tr>
<th>Subject Name</th>
<th>Subject Index</th>
<th>Subject Type</th>
<th>Bound</th>
<th>Limit</th>
<th>Scale</th>
</tr>
</thead>
<tbody>
<tr>
<td>st</td>
<td>1</td>
<td>IResp</td>
<td>Lower</td>
<td>25.000000</td>
<td>25.000000</td>
</tr>
</tbody>
</table>

DESIGN OBJECTIVE INFORMATION

<table>
<thead>
<tr>
<th>Subject Name</th>
<th>Subject Index</th>
<th>Subject Type</th>
<th>Worst Value</th>
<th>Weight Factor</th>
<th>Target</th>
</tr>
</thead>
<tbody>
<tr>
<td>mflx</td>
<td>0</td>
<td>IResp</td>
<td>Undefined</td>
<td>1.000000</td>
<td>Maximize</td>
</tr>
</tbody>
</table>

LINK INFORMATION

<table>
<thead>
<tr>
<th>ID</th>
<th>Name</th>
<th>Constant</th>
<th>Multipliers</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Lnk_l</td>
<td>0.00</td>
<td>1</td>
</tr>
</tbody>
</table>
## DESIGN POINT 0

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>d</td>
<td>IDVar</td>
<td>2.500000</td>
</tr>
<tr>
<td>h</td>
<td>IDVar</td>
<td>2.500000</td>
</tr>
<tr>
<td>H</td>
<td>IDVar</td>
<td>5.000000</td>
</tr>
<tr>
<td>lref</td>
<td>LDVar</td>
<td>2.500000</td>
</tr>
<tr>
<td>mflx</td>
<td>IResp</td>
<td>1.103041</td>
</tr>
<tr>
<td>st</td>
<td>IResp</td>
<td>11.71658</td>
</tr>
</tbody>
</table>

---

### FINAL RESULTS FOR ITERATION 0

- **Best Design Point:** 0
- **Best Objective:** 1.103041
- **Worst Constraint:** 0.5313368

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Lower Bound</th>
<th>Value</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>d</td>
<td>IDVar</td>
<td>0.2500000</td>
<td>2.5000000</td>
<td>5.000000</td>
</tr>
<tr>
<td>h</td>
<td>IDVar</td>
<td>0.2500000</td>
<td>2.5000000</td>
<td>5.000000</td>
</tr>
<tr>
<td>H</td>
<td>IDVar</td>
<td>0.7500000</td>
<td>5.000000</td>
<td>10.00000</td>
</tr>
<tr>
<td>lref</td>
<td>LDVar</td>
<td>-1.000000e+30</td>
<td>2.5000000</td>
<td>1.000000e+30</td>
</tr>
</tbody>
</table>

### OBJECTIVE FUNCTION(S)

- **mflx IResp**
  - Target: maximize
  - Worst Value: 1.103
  - Objective Value: 1.103041

### CONSTRAINED RESPONSES

- **st IResp**
  - U/L: L
  - Value: 11.71658
  - Scaled Value: 0.4686632
  - Bound: 25.00000

* - active bound   V - violated bound

---

### FINAL RESULTS FOR ITERATION 4

- **Best Design Point:** 8
- **Best Objective:** 0.8771339
- **Worst Constraint:** 0.0241309

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Lower Bound</th>
<th>Value</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>d</td>
<td>IDVar</td>
<td>0.2500000</td>
<td>1.472216</td>
<td>5.000000</td>
</tr>
<tr>
<td>h</td>
<td>IDVar</td>
<td>0.2500000</td>
<td>0.2500000</td>
<td>5.000000</td>
</tr>
<tr>
<td>H</td>
<td>IDVar</td>
<td>0.7500000</td>
<td>3.314614</td>
<td>10.00000</td>
</tr>
<tr>
<td>lref</td>
<td>LDVar</td>
<td>-1.000000e+30</td>
<td>1.472216</td>
<td>1.000000e+30</td>
</tr>
</tbody>
</table>

### OBJECTIVE FUNCTION(S)

- **mflx IResp**
  - Target: maximize
  - Worst Value: 1.103
  - Objective Value: 1.103041

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
### DESIGN VARIABLES

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Lower Bound</th>
<th>Value</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>d</td>
<td>IDVar</td>
<td>0.2500000</td>
<td>1.822471</td>
<td>5.000000</td>
</tr>
<tr>
<td>h</td>
<td>IDVar</td>
<td>0.2500000</td>
<td>2.118450</td>
<td>5.000000</td>
</tr>
<tr>
<td>H</td>
<td>IDVar</td>
<td>0.7500000</td>
<td>2.117358</td>
<td>10.00000</td>
</tr>
<tr>
<td>lref</td>
<td>LDVar</td>
<td>-1.000000e+30</td>
<td>1.822471</td>
<td>1.000000e+30</td>
</tr>
</tbody>
</table>

### OBJECTIVE FUNCTION(S)

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Target Value</th>
<th>Worst DVar/Response Value</th>
<th>Objective Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>mflx</td>
<td>IResp</td>
<td>maximize</td>
<td>1.103</td>
<td>1.745752</td>
</tr>
</tbody>
</table>

### CONSTRAINED RESPONSES

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>U/L</th>
<th>Value</th>
<th>Scaled Value</th>
<th>Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>st</td>
<td>IResp</td>
<td>L</td>
<td>25.18823</td>
<td>1.007529</td>
<td>25.00000 *</td>
</tr>
</tbody>
</table>

* - active bound  V - violated bound

---

**DESIGN HISTORY**

<table>
<thead>
<tr>
<th>Iter #</th>
<th>Best Objective</th>
<th>Worst Constraint</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1.10304117</td>
<td>0.531336838</td>
</tr>
<tr>
<td>1</td>
<td>1.37231493</td>
<td>0.274487834</td>
</tr>
<tr>
<td>2</td>
<td>1.73555470</td>
<td>0.178456259</td>
</tr>
<tr>
<td>3</td>
<td>0.877133906</td>
<td>0.0241308990</td>
</tr>
<tr>
<td>4</td>
<td>0.877133906</td>
<td>0.0241308990</td>
</tr>
<tr>
<td>5</td>
<td>0.877133906</td>
<td>0.0241308990</td>
</tr>
<tr>
<td>6</td>
<td>1.74488604</td>
<td>-0.00761926512</td>
</tr>
<tr>
<td>7</td>
<td>1.74575222</td>
<td>-0.00752938238</td>
</tr>
</tbody>
</table>

---

**MESSAGE SUMMARY (Task ID: 12)**
VisualDOC task report of the cavity shape optimization with Bezier curves in Section 7.2.2 (Table 7.5) is given below. In this report, the sub iteration results are not included. Moreover only the final results for initial, fifth and the last iterations are given due to their similarity.

**VisualDOC Task Report**

```
  * Report Version : 3.00 (0)
  * Database Version : 3.00 (0)
  * Database File Name : /.../lxpnt
  * Task Number : 6
  * Task Name : lfxpnt-lref
  * Run Status : Completed
  * Run Started On : Mon Apr 2 18:10:12 2007
```

**DESIGN TASK CONTROL INFORMATION**

```
1lxpnt-lref
Nonlinear, Constrained Optimization
```

```
Task Database ID : 6
Task Created On : Sun Apr 1 01:22:37 2007
Optimization Method : SQP
Print Control : None
Objective : Maximize
Constraint Tolerance : -0.03
Violated Constraint Tolerance : 0.003
```
Gradients Calculated By: First Forward Difference
Relative Finite Difference Step: 0.1
Minimum Finite Difference Step: 0.01

--- Optimizer Parameters
Relative Hard Convergence Criteria: 0.001
Absolute Objective Convergence Tol.: Default
Consecutive Iter. for Convergence: 2
Iter. b/w Design Variable Scaling: Default
Max. Number of Iter. by the Optimizer: 100

--- SLP/SQP Sub-Problem
Maximum Iterations: 50
Consecutive Iter. for Convergence: 2

--- Independent Design Variables: 6
Linked Design Variables: 2
Synthetic Design Variables: 0
Constant Design Variables: 0
Independent Responses: 8
Linked Responses: 0
Synthetic Responses: 0
Constraints: 7
Objectives: 1
Link Specifications: 1
Number of Discrete Set Specifications: 0

Analysis Program: /.../lfxpnt.pl
Design Variable Transfer: dvar.vef
Responses Transfer: resp.vef

--- DESIGN VARIABLE INFORMATION

### Transfer Sequence
<table>
<thead>
<tr>
<th>Transfer Sequence</th>
<th>Type</th>
<th>Name</th>
<th>DB ID</th>
<th>Auxil DB ID</th>
<th>Lower Bound</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>IDVar</td>
<td>x1</td>
<td>1</td>
<td></td>
<td>-15.00000</td>
<td>5.000000</td>
</tr>
<tr>
<td>2</td>
<td>IDVar</td>
<td>y1</td>
<td>2</td>
<td></td>
<td>1.000000</td>
<td>4.000000</td>
</tr>
<tr>
<td>3</td>
<td>IDVar</td>
<td>x2</td>
<td>3</td>
<td></td>
<td>-15.00000</td>
<td>5.000000</td>
</tr>
<tr>
<td>4</td>
<td>IDVar</td>
<td>y2</td>
<td>4</td>
<td></td>
<td>-5.000000</td>
<td>4.000000</td>
</tr>
<tr>
<td>5</td>
<td>IDVar</td>
<td>x3</td>
<td>5</td>
<td></td>
<td>-2.000000</td>
<td>-0.1000000</td>
</tr>
<tr>
<td>6</td>
<td>IDVar</td>
<td>y3</td>
<td>6</td>
<td></td>
<td>1.000000</td>
<td>8.000000</td>
</tr>
<tr>
<td>7</td>
<td>LDVar</td>
<td>lref</td>
<td>7</td>
<td>1</td>
<td>-1.000000e+30</td>
<td>1.000000e+30</td>
</tr>
<tr>
<td>8</td>
<td>LDVar</td>
<td>lref2</td>
<td>8</td>
<td>1</td>
<td>-1.000000e+30</td>
<td>1.000000e+30</td>
</tr>
</tbody>
</table>

--- LINKED DESIGN VARIABLES

<table>
<thead>
<tr>
<th>DVar Name</th>
<th>Link Name</th>
<th>Arguments</th>
</tr>
</thead>
<tbody>
<tr>
<td>lref</td>
<td>Lnk_5</td>
<td>x3</td>
</tr>
<tr>
<td>lref2</td>
<td>Lnk_5</td>
<td>x3</td>
</tr>
</tbody>
</table>

--- DESIGN RESPONSE INFORMATION

<table>
<thead>
<tr>
<th>Transfer Sequence</th>
<th>Type</th>
<th>Name</th>
<th>DB ID</th>
<th>Auxil DB ID</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>IResp</td>
<td>mflx</td>
<td>1</td>
<td>none</td>
</tr>
<tr>
<td>2</td>
<td>IResp</td>
<td>st</td>
<td>2</td>
<td>none</td>
</tr>
<tr>
<td>3</td>
<td>IResp</td>
<td>rxxr</td>
<td>3</td>
<td>none</td>
</tr>
<tr>
<td>4</td>
<td>IResp</td>
<td>dmin</td>
<td>4</td>
<td>none</td>
</tr>
</tbody>
</table>
### DESIGN CONSTRAINT INFORMATION

<table>
<thead>
<tr>
<th>Subject Name</th>
<th>Index</th>
<th>Type</th>
<th>Bound</th>
<th>Limit</th>
<th>Scale Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>st</td>
<td>1</td>
<td>IResp</td>
<td>Lower</td>
<td>25.00000</td>
<td>25.00000</td>
</tr>
<tr>
<td>rxry</td>
<td>2</td>
<td>IResp</td>
<td>Upper</td>
<td>5.00000</td>
<td>5.00000</td>
</tr>
<tr>
<td>dmin</td>
<td>3</td>
<td>IResp</td>
<td>Upper</td>
<td>-0.0100000</td>
<td>1.00000</td>
</tr>
<tr>
<td>amaxtheta</td>
<td>4</td>
<td>IResp</td>
<td>Upper</td>
<td>271.0000</td>
<td>271.0000</td>
</tr>
<tr>
<td>dydx</td>
<td>5</td>
<td>IResp</td>
<td>Lower</td>
<td>0.1000000</td>
<td>1.0000000</td>
</tr>
<tr>
<td>ymnall</td>
<td>6</td>
<td>IResp</td>
<td>Lower</td>
<td>0.0000000</td>
<td>1.0000000</td>
</tr>
<tr>
<td>ymin</td>
<td>7</td>
<td>IResp</td>
<td>Lower</td>
<td>0.2000000</td>
<td>1.0000000</td>
</tr>
</tbody>
</table>

### DESIGN OBJECTIVE INFORMATION

<table>
<thead>
<tr>
<th>Subject Name</th>
<th>Index</th>
<th>Type</th>
<th>Worst Value</th>
<th>Weight Factor</th>
<th>Target</th>
</tr>
</thead>
<tbody>
<tr>
<td>mflx</td>
<td>0</td>
<td>IResp</td>
<td>Undefined</td>
<td>1.000000</td>
<td>Maximize</td>
</tr>
</tbody>
</table>

### LINK INFORMATION

<table>
<thead>
<tr>
<th>ID</th>
<th>Name</th>
<th>Constant</th>
<th>Multipliers</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Lnk_5</td>
<td>0.00</td>
<td>-2</td>
</tr>
</tbody>
</table>

### DESIGN POINT 0

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>x1</td>
<td>IDVar</td>
<td>-9.000000</td>
</tr>
<tr>
<td>y1</td>
<td>IDVar</td>
<td>3.500000</td>
</tr>
<tr>
<td>x2</td>
<td>IDVar</td>
<td>0.5000000</td>
</tr>
<tr>
<td>y2</td>
<td>IDVar</td>
<td>3.000000</td>
</tr>
<tr>
<td>x3</td>
<td>IDVar</td>
<td>-0.5000000</td>
</tr>
<tr>
<td>y3</td>
<td>IDVar</td>
<td>4.000000</td>
</tr>
<tr>
<td>lref</td>
<td>LDVar</td>
<td>1.0000000</td>
</tr>
<tr>
<td>lref2</td>
<td>LDVar</td>
<td>1.0000000</td>
</tr>
<tr>
<td>mflx</td>
<td>IResp</td>
<td>0.70243944</td>
</tr>
<tr>
<td>st</td>
<td>IResp</td>
<td>43.68055</td>
</tr>
<tr>
<td>rxry</td>
<td>IResp</td>
<td>0.2697400</td>
</tr>
<tr>
<td>dmin</td>
<td>IResp</td>
<td>-0.4263800</td>
</tr>
<tr>
<td>amaxtheta</td>
<td>IResp</td>
<td>223.6700</td>
</tr>
<tr>
<td>dydx</td>
<td>IResp</td>
<td>3.052800</td>
</tr>
<tr>
<td>ymnall</td>
<td>IResp</td>
<td>0.05246100</td>
</tr>
<tr>
<td>ymin</td>
<td>IResp</td>
<td>2.427300</td>
</tr>
</tbody>
</table>

### FINAL RESULTS FOR ITERATION 0

- Best Design Point: 0
- Best Objective: 0.7024394
- Worst Constraint: -0.05246100

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
DESIGN VARIABLES

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Lower Bound</th>
<th>Value</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>x1</td>
<td>IDVar</td>
<td>-15.00000</td>
<td>-9.00000</td>
<td>5.000000</td>
</tr>
<tr>
<td>y1</td>
<td>IDVar</td>
<td>1.000000</td>
<td>3.50000</td>
<td>4.000000</td>
</tr>
<tr>
<td>x2</td>
<td>IDVar</td>
<td>-15.00000</td>
<td>0.500000</td>
<td>5.000000</td>
</tr>
<tr>
<td>y2</td>
<td>IDVar</td>
<td>-5.000000</td>
<td>3.000000</td>
<td>4.000000</td>
</tr>
<tr>
<td>x3</td>
<td>IDVar</td>
<td>-2.000000</td>
<td>-0.500000</td>
<td>-0.100000</td>
</tr>
<tr>
<td>y3</td>
<td>IDVar</td>
<td>1.000000</td>
<td>4.000000</td>
<td>8.000000</td>
</tr>
<tr>
<td>rref</td>
<td>LDVar</td>
<td>-1.000000e+30</td>
<td>1.000000</td>
<td>1.000000e+30</td>
</tr>
<tr>
<td>rref2</td>
<td>LDVar</td>
<td>-1.000000e+30</td>
<td>1.000000</td>
<td>1.000000e+30</td>
</tr>
</tbody>
</table>

OBJECTIVE FUNCTION(S)

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Target Value</th>
<th>Worst Value</th>
<th>DVar/Response</th>
<th>Objective Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>mflx</td>
<td>IResp</td>
<td>maximize</td>
<td>0.7024</td>
<td>2.685344</td>
<td>2.685344</td>
</tr>
</tbody>
</table>

CONSTRAINED RESPONSES

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>U/L Value</th>
<th>Value</th>
<th>Scaled Value</th>
<th>Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>st</td>
<td>IResp</td>
<td>L</td>
<td>25.56269</td>
<td>1.022508</td>
<td>25.00000 *</td>
</tr>
</tbody>
</table>
### DESIGN VARIABLES

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Lower Bound</th>
<th>Value</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>x1 IDVar</td>
<td></td>
<td>-15.00000</td>
<td>-7.552080</td>
<td>5.000000</td>
</tr>
<tr>
<td>y1 IDVar</td>
<td></td>
<td>1.00000</td>
<td>2.292279</td>
<td>4.000000</td>
</tr>
<tr>
<td>x2 IDVar</td>
<td></td>
<td>-15.00000</td>
<td>0.06153782</td>
<td>5.000000</td>
</tr>
<tr>
<td>y2 IDVar</td>
<td></td>
<td>-5.00000</td>
<td>-0.2738694</td>
<td>4.000000</td>
</tr>
<tr>
<td>x3 IDVar</td>
<td></td>
<td>-2.00000</td>
<td>-1.146870</td>
<td>-0.100000</td>
</tr>
<tr>
<td>y3 IDVar</td>
<td></td>
<td>1.00000</td>
<td>6.787092</td>
<td>8.000000</td>
</tr>
<tr>
<td>lref LDVar</td>
<td></td>
<td>-1.000000e+30</td>
<td>2.293741</td>
<td>1.0000000e+30</td>
</tr>
<tr>
<td>lref2 LDVar</td>
<td></td>
<td>-1.000000e+30</td>
<td>2.293741</td>
<td>1.0000000e+30</td>
</tr>
</tbody>
</table>

### OBJECTIVE FUNCTION(S)

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Value</th>
<th>Value</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>mflx IResp</td>
<td>maximize</td>
<td>0.7024</td>
<td>2.685344</td>
<td>2.685344</td>
</tr>
</tbody>
</table>

### CONSTRAINED RESPONSES

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>U/L</th>
<th>Value</th>
<th>Scaled Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>st IResp</td>
<td>L</td>
<td></td>
<td>25.56269</td>
<td>1.022508</td>
</tr>
</tbody>
</table>

* - active bound  V - violated bound

### DESIGN HISTORY

<table>
<thead>
<tr>
<th>Iter #</th>
<th>Best Objective</th>
<th>Worst Constraint</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.702439427</td>
<td>-0.052461000</td>
</tr>
<tr>
<td>1</td>
<td>1.70472181</td>
<td>-0.042535000</td>
</tr>
<tr>
<td>2</td>
<td>2.44736981</td>
<td>-0.031478000</td>
</tr>
<tr>
<td>3</td>
<td>2.46721721</td>
<td>-0.030237000</td>
</tr>
<tr>
<td>4</td>
<td>2.51181006</td>
<td>-0.033259000</td>
</tr>
<tr>
<td>5</td>
<td>2.68534350</td>
<td>-0.0225077127</td>
</tr>
<tr>
<td>6</td>
<td>2.68534350</td>
<td>-0.0225077127</td>
</tr>
<tr>
<td>7</td>
<td>2.68534350</td>
<td>-0.0225077127</td>
</tr>
<tr>
<td>8</td>
<td>2.68534350</td>
<td>-0.0225077127</td>
</tr>
<tr>
<td>9</td>
<td>2.68534350</td>
<td>-0.0225077127</td>
</tr>
<tr>
<td>10</td>
<td>2.68534350</td>
<td>-0.0225077127</td>
</tr>
<tr>
<td>11</td>
<td>2.68534350</td>
<td>-0.0225077127</td>
</tr>
</tbody>
</table>

### RESULTS SUMMARY

- Run Started On: Mon Apr 2 18:10:12 2007
- Warning Messages: 0
- Error Messages: 0
- Total Analysis Calls: 95
- Total Gradient Requests: 11
- Total # Points in the Design Space: 38
- Iteration of the Best Design (cont.): 5
- Best Objective (cont.): 2.685344
- Worst Constraint (cont.): -0.02250771
- Continuous Run Status: Completed
Continuous Stop Code : null search direction in the QP sub-problem

Timing (seconds)
Analysis : 151357.66
Transfer :  0.74
Database :   5.01
Total : 151368.32

=================================================================
VITA

Mehti Koklu was born on May 20, 1976 in Kahramanmaras, Turkey. He earned his high school degree in 1995 from Gaziantep Science High School, a prestigious high school institution that selects its students through a nation wide exam and provides a sophisticated education in sciences and mathematics. After an annual nationwide university entrance exam, he was ranked in the first top percentile among the 1.5 million students and admitted to Aeronautical Engineering at Istanbul Technical University (ITU) in 1995. Throughout his undergraduate study, he was awarded with a four-year, full scholarship from the Turkish Education Foundation. He received his Bachelor of Science degree in Aeronautical Engineering at Istanbul Technical University in June 1999. Immediately thereafter, he was appointed as a research assistant in the Aeronautical Engineering department of the same university while conducting his graduate studies. After the completion of the Master of Science degree in June 2001, he was admitted into Aerospace Engineering Department at Old Dominion University in January 2003 to pursue his Ph.D. study. During his Ph.D. study, he was awarded a research assistantship by the same department. He was also awarded the “Computational Engineering Doctoral Fellowship” by the College of Engineering and Technology at Old Dominion University during 2004-2005 and the “University Dissertation Fellowship” by Old Dominion University during 2006-2007. He was involved in research work pertaining to CFD, Design Optimization, and MEMS. His papers have appeared in several journals and conference proceedings.

On 20 September 2003, he married Emine Bay and now he is the father of an 8-month-old boy, Bera E. Koklu.